Objectives for the National Instruments Multisim Course

Introduction

Course Objectives and Overview

This course will provide you with an introduction to Multisim's many features. At the end of the three hours, you should have a basic understanding of the user interface (UI), schematic capture and simulation. You will also see how Multisim fits into the broader PCB design flow (Figure 1 and Figure 2).

The course is split into sections, each containing associated exercises to help solidify the concepts presented. There is also an Appendix, which provides some direction on how to integrate Multisim with National Instruments' products to create a truly integrated benchtop platform for design and verification.

Course Materials

In order to successfully complete this course, you will need the following:

- Multisim Software.
- Multisim User Guide and helpfiles (available in soft copies installed with the software).
- Associated circuits (associated circuits found on the CD in the back of this book).

The Electronics Workbench Portfolio
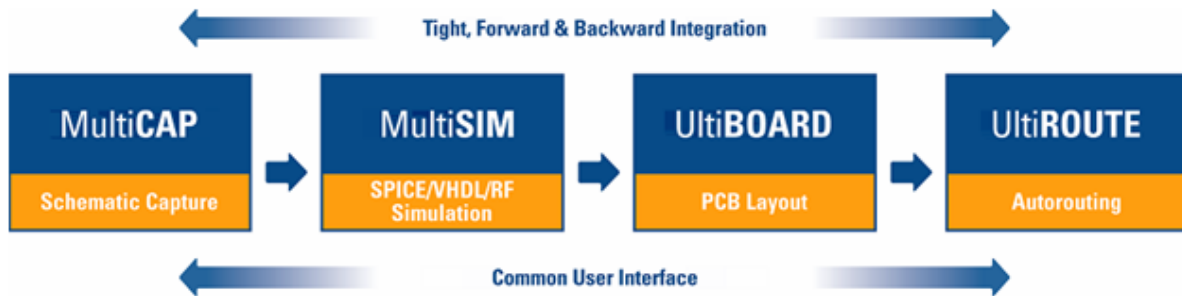
Electronics Workbench Product Flow

Multicap

Multicap 9 is the industry's most intuitive and powerful schematic capture program. Multicap's innovative and timesaving features, including modeless operation, powerful autowiring and a comprehensive database organized into logical parts bins on your desktop, allow designs to be captured almost as fast as they can be conceptualized. Repetitive tasks are optimized, leaving you freer to create, test and ultimately perfect your designs, resulting in superior products and minimizing time-to-market.

Multisim

Multisim, the world's only interactive circuit simulator, allows you to design better products in less time. Multisim includes a completely integrated version of Multicap, making it the ideal tool for capturing schematics and then instantly simulating circuits.

Multisim 9 also offers integration with National Instruments LabVIEW and SignalExpress, allowing you to tightly integrate design and test.

Ultiboard

Ultiboard has been carefully designed to maximize your productivity. By optimizing the most common repetitive tasks such as part and trace placement, the number of keystrokes and mouse movements required to lay out any design has been dramatically reduced.

Ultiboard handles today's higher speed designs with ease using constraint driven layout. Innovative features such as Real-Time design rule checking, "Push & Shove" components & traces, component nudging with trace rubberbanding, "Follow-me" trace editing and an Automatic Connection machine ensure that you rapidly complete an error-free board.

Ultiroute for Professional Customers

For all but the most basic PCBs, the density and complexity of today's designs make manual component and trace placement techniques impractical. For these designs, Electronics Workbench offers Ultiroute — a state-of-the-art autorouting and autoplacement tool.

Ultiroute ensures high circuit performance and lower production costs for these demanding projects by uniquely combining the best of gridless (shape-based) and grid-based routing.

Integrated Design and Validation with National Instruments

The Integrated Design Flow

LabVIEW

NI LabVIEW is the graphical development environment for creating flexible and scalable test, measurement, and control applications rapidly and at minimal cost. With LabVIEW, engineers and scientists interface with real-world signals, analyze data for meaningful information, and share results and applications. Regardless of experience, LabVIEW makes development fast and easy for all users.

SignalExpress

SignalExpress is interactive software for quickly acquiring, comparing, automating, and storing measurements. Use SignalExpress to streamline your exploratory and automated measurement tasks for electronics design, validation, and test.

SignalExpress introduces an innovative approach to configuring your measurements using intuitive drag-and-drop steps that do not require code development.

NI ELVIS

National Instruments Educational Laboratory Virtual Instrumentation Suite (NI ELVIS) is a LabVIEW-based design and prototyping environment for university science and engineering laboratories. NI ELVIS consists of LabVIEW-based virtual instruments, a multifunction data acquisition device, and a custom-designed benchtop workstation and prototyping board. This combination provides a ready-to-use suite of instruments found in all educational laboratories. Because it is based on LabVIEW and provides complete data acquisition and prototyping capabilities, the system is ideal for academic coursework, from lower-division classes to advanced project-based curricula. Curriculum applications include electronics design,

communications, controls, mechatronics, instrumentation, and data acquisition.

Schematic Capture in MultiSim

The Benefits of Integrated Capture and Simulation

Multisim provides users with the unique ability to capture and simulate from within the very same integrated environment. The advantages of this approach are many. Users new to Multisim do not have to worry about sophisticated SPICE (Simulation Program with Integrated Circuit Emphasis) syntax and commands, while advanced users have easy access to all SPICE details.

Multisim makes capturing schematics easier and more intuitive than ever. A spreadsheet view allows users to easily modify characteristics of any number of components simultaneously: from PBC footprint to SPICE model. Modeless operation provides the most efficient way of placing components and wiring them together. Working with both analog and digital multisection components is intuitive and simple.

In addition to traditional SPICE analyses, Multisim allows users to intuitively connect virtual instruments to their schematics. Virtual instruments make it fast and easy to view interactive simulation results by replicating the real-world environment.

Multisim also provides special components known as interactive parts which can be modified while a simulation is running. Interactive parts such as switches and potentiometers, will immediately and accurately affect the results of simulation.

When the need arises for more advanced analysis, Multisim delivers over 15 sophisticated analyses. Some examples of analyses include AC, Monte Carlo, Worst Case, and Fourier. Provided natively within the Multisim environment is a powerful Grapher, which allows the customized viewing of simulation data and analyses.

The integrated capture and simulation environment provided by Multisim is a natural fit for any circuit designer, and will save both time and frustration throughout the entire circuit design process.

The Multisim Environment

Introducing the Multisim Environment

Multisim's user interface consists of the basic elements, illustrated in Figure 1 below.
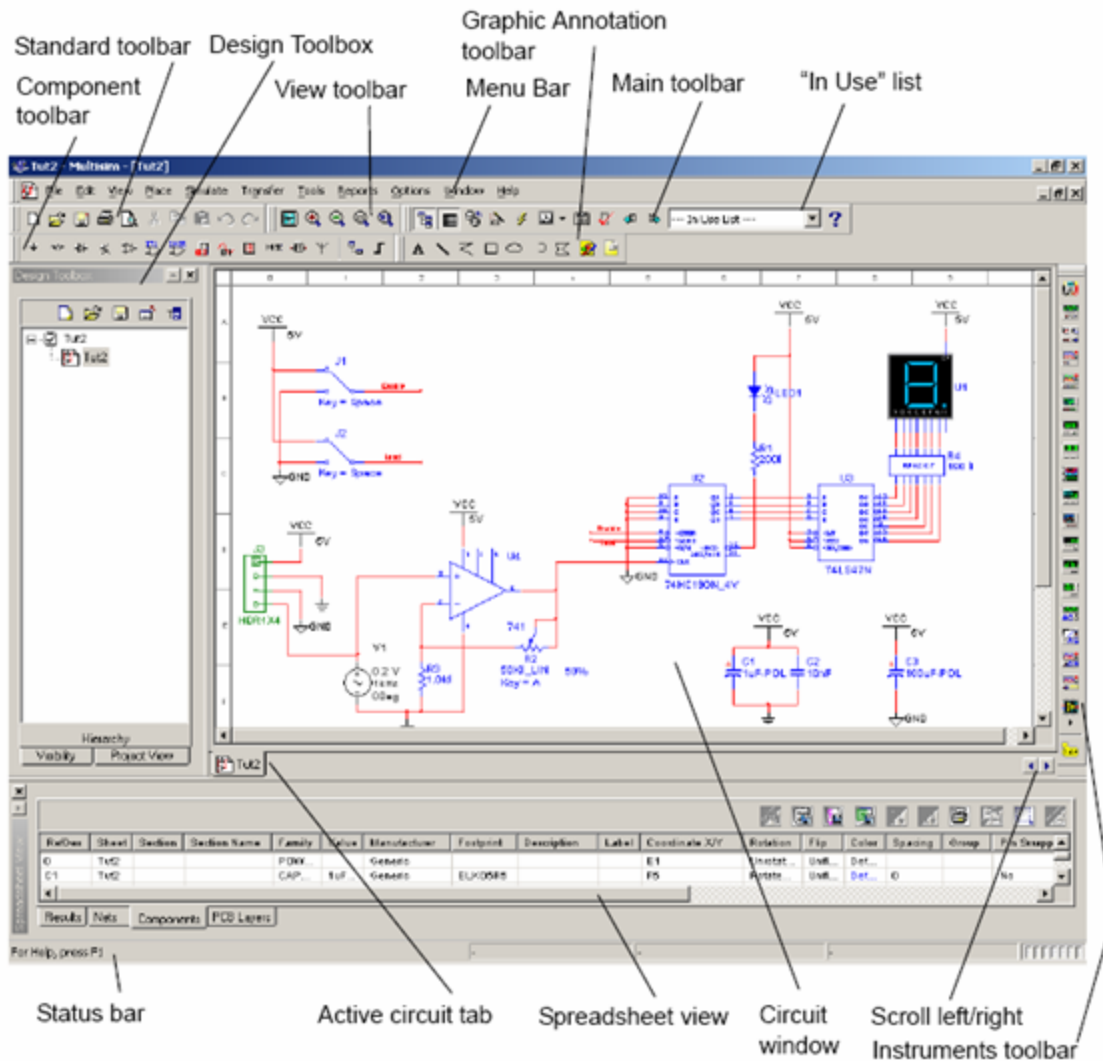
The MultiSim Environment

The Design Toolbox

The Design Toolbox is used to manage various elements in the schematic. The Visibility tab lets you choose which layers to display on the current sheet on the workspace. The Hierarchy tab contains a tree that shows the dependencies of the files in the design that you have open. The Project tab

displays information about the current project. Users can add files to the existing folders of the current project, control access to files, and archive designs.

Configuring global options

Global options allow users to configure specifics of the Multisim environment. They can be modified by accessing the preferences dialog box. Choose Options/Global Preferences. The Preferences dialog box appears, offering you the following tabs:

• Paths—where you can change the filepaths for the databases and other settings.

• Save—where you set up Auto-backup timing and whether you want to save simulation data with instruments.

• Parts—where you set up component placement mode and the symbol standard (ANSI or DIN). You also set up default digital simulation settings.

• General—where you set up selection rectangle behavior, mouse wheel behavior, bus wiring and auto-wiring behavior.
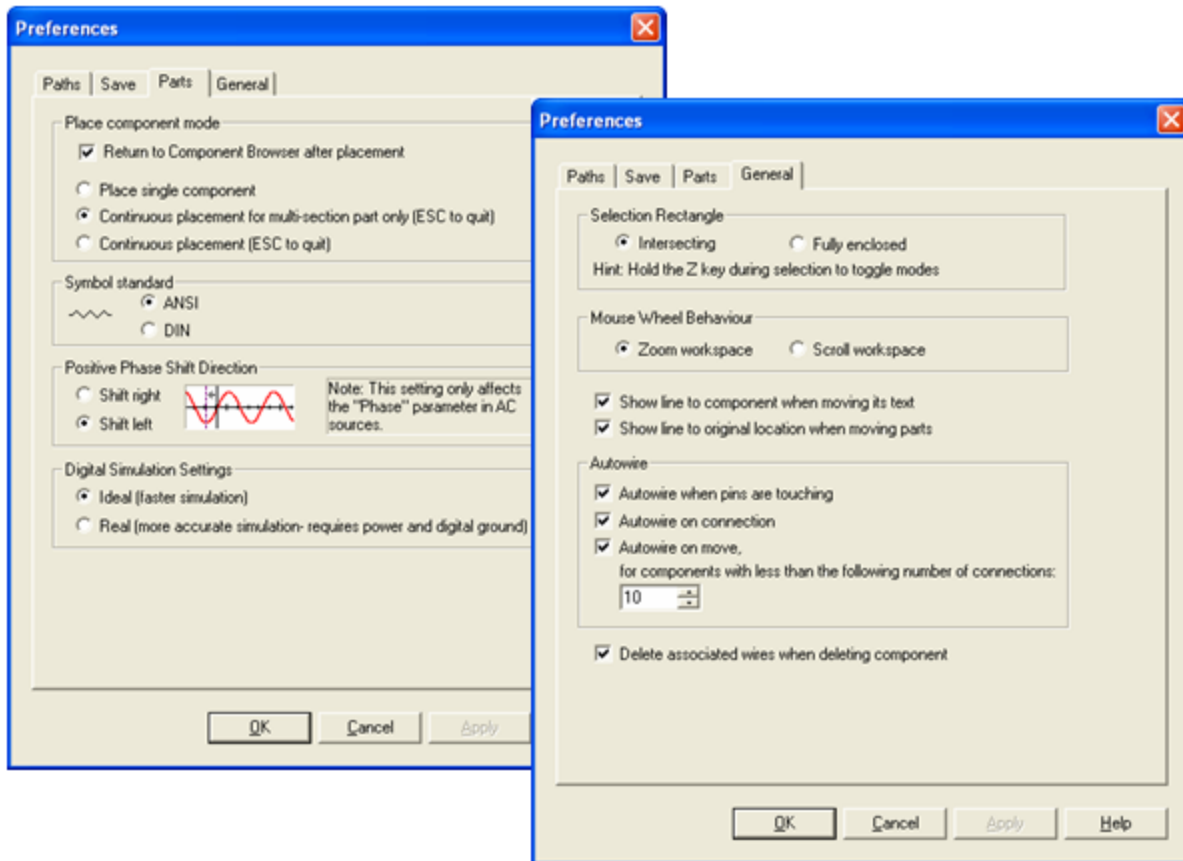
Various Global Preferences

Configuring sheet options

The Sheet Properties dialog box is used to set up the preferences for each sheet. These preferences are saved with the circuit file so that if the circuit is opened on another computer, it will use the same settings.

The sheet specific settings are arranged in the following tabs:

• Circuit—where you set the color scheme and the display properties of workspace text.

• Workspace—where you set the sheet size and properties.

• Wiring—where you set the wire and bus options.

• Font—where you select the font, font size and style for text elements on the circuit.

• PCB—where you set up options for your printed circuit board.

• Visibility—where you enable and disable custom annotation layers.

Consult the Multisim User Guide or Multisim helpfile for a detailed description of each sheet property.

Sheet properties can be applied to a single schematic, or saved for all future circuits.
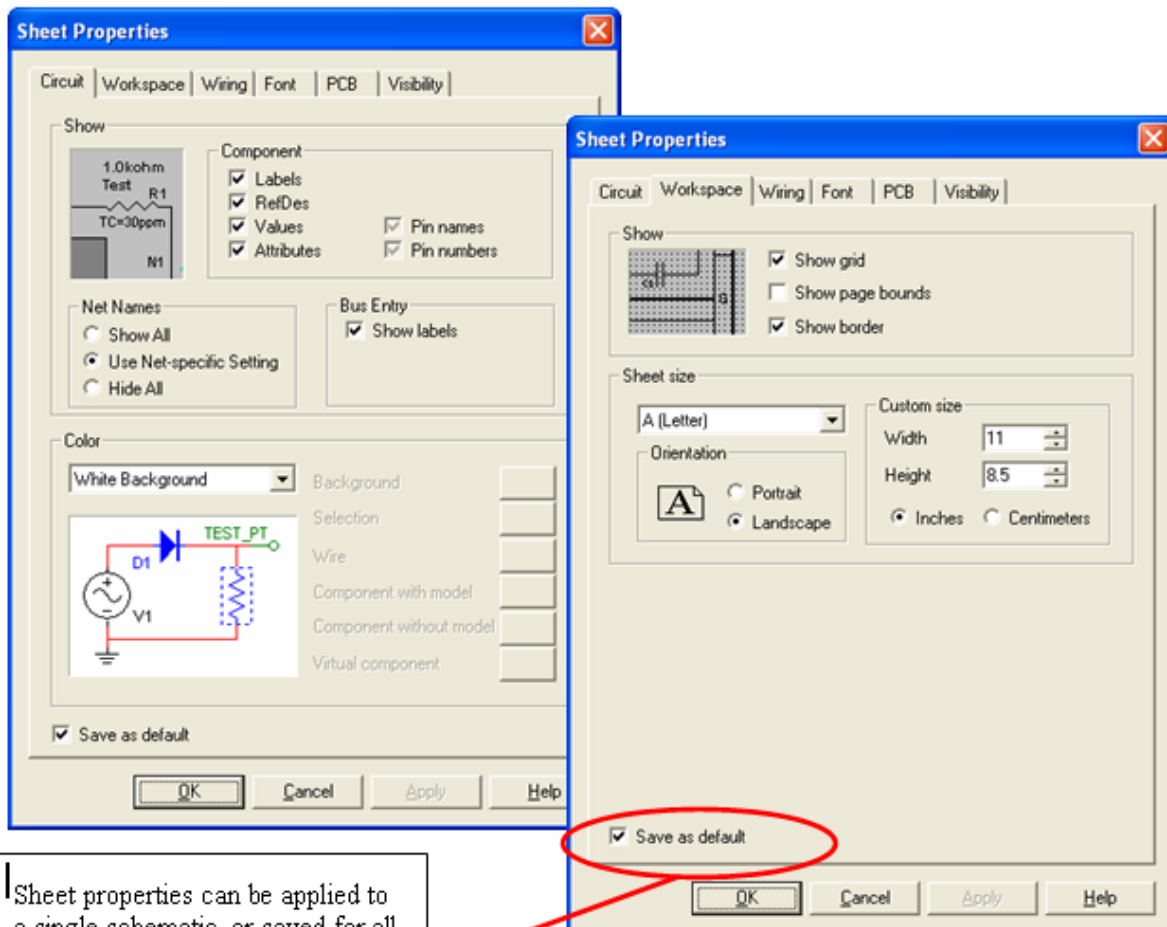
Various Sheet Properties

Customizing the user interface

The Multisim user interface is highly customizable. Customizations are context sensitive. Toolbars may be docked in various positions and orientations. The contents of the toolbars may be customized. New toolbars may be created. The menu system is fully customizable, including all pop-up menus for the various object types.

As well, the keyboard shortcut system is customizable. This allows for key combinations to be assigned to any command that may be placed in a menu or on a toolbar.

Note: To avoid interfering with interactive simulation, it is recommended that users only assign commands to key combinations (such as Ctrl-E).

For example, the toolbars and docking windows may be re-configured as you move from a circuit sheet to a description sheet.

To customize the user interface, select Options/Customize User Interface. Using the Customize dialog, users can create or modify toolbars, assign or change keyboard shortcuts, customize and create menus, and modify the way the user interface appears.
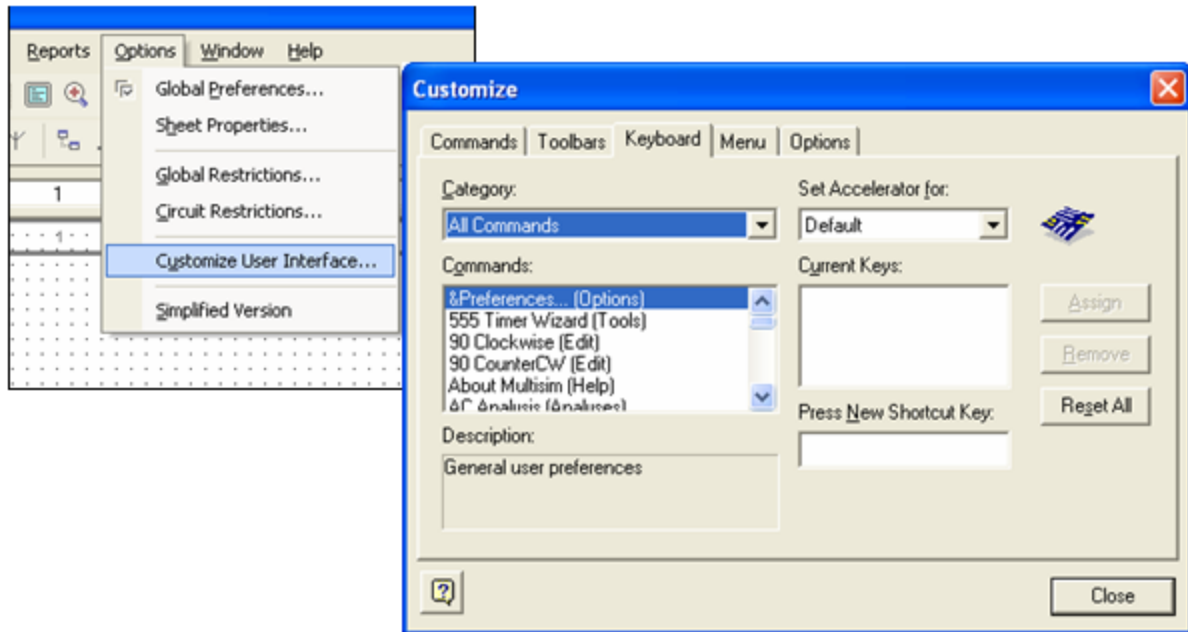
Customize Dialog Box

Exercise: Introducing the National Instruments Multisim Environment

Approximate time to complete: 10 minutes

The purpose of this exercise is to familiarize yourself with the Multisim environment. You will have the chance to explore the sheet and global options, as well as examine the various toolbars and menu items available.

Objectives

- Become familiar with the general Multisim interface.
- Become familiar with configuring Multisim workspace parameters.

Procedure

1. Launch Multisim

    1. Select File/OpenSamples, and open [AMPMOD.ms9](#).
    2. Experiment with the different views that Multisim provides:

        1. Select View/Spreadsheet to toggle the spreadsheet view.
        2. Browse the Nets, Components,and PCB Layerstabs.
        3. How many uniquely numbered nets are there?_____

    3. Select View/Circuit Description Box.This is where designers can view detailed information about their design. To edit the contents, select Tools/Description Box Editor.
    4. Select View/Design Toolbox.This view gives designers insight into the files, sub-circuits and other elements of a given design.

2. Experiment with Global Preferences and Sheet Properties.

    1. Select Options/Sheet Properties.

        1. Try toggling on and off the grid in the Workspace tab (to view the changes, click OK or Apply).

2. Try switching the colors of the environment in the Circuittab (to view the changes, click OK or Apply).

2. Select Options/Global Preferences.

1. Enable the Auto-backup feature in the Save tab.
2. Toggle on or off the Return to Component Browser check box in the Parts tab according to your personal preference.

3. Examine the settings under the Generaltab. What is the default Selection Rectangle mode?
4. If time permits, continuing experimenting with the Multisim environment. Try to place an arbitrary component onto the schematic.
5. Close the schematic by clicking File/Close.

End of Exercise

AMPMOD.ms9

Electrical and Electronic Components Available in National Instruments Multisim

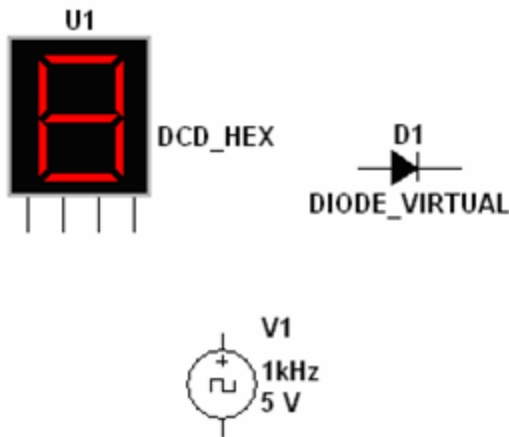# Components

Components Overview

Components comprise the basis for any schematic. A component is any part that can be placed onto the schematic. Multisim defines two broad categories of parts: real and virtual. It is important to understand the difference between these parts, in order to fully utilize their advantages.

Real components can be differentiated from virtual parts because real components have a specific value that cannot be changed, and a PCB footprint.

Virtual components are simulation-only components, which can be assigned user-defined characteristics. For example, a virtual resistor can take on any resistance (such as 3.86654 Ohms). Virtual components help designers to check calculations by simulating designs with precise component values. Virtual components can also be idealized components such as the 4-pin Hex display shown in Figure 1.

Multisim also provides other classifications of components: analog, digital, mixed-mode, animated, interactive, multi-section digital, electromechanical, and radio-frequency (RF) components.
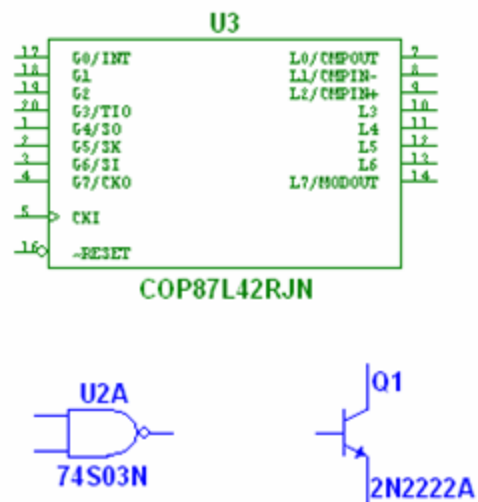
Various Component Symbols: 7-Segment Display, Diode D1, Voltage Source V1, NAND gate U2A, Microcontroller U3 and Transistor Q1

Interactive Components

Multisim provides a method for interacting with certain components that are placed on the schematic. Changes to these components will affect the simulation results on-the-fly. Components are controlled by pressing the key listed beside the component.

For the components shown in Figure 2 below, pressing the A keywill increase the resistance of the potentiometer toward 100% of the shown value (1kΩ); to decrease the resistance, hold the Shift key then press the A key. Pressing the Spacebar will toggle the switch to be either closed or open.

The keyboard shortcut key can be changed by double-clicking on the component, and choosing the desired key from the drop-down box as shown below.
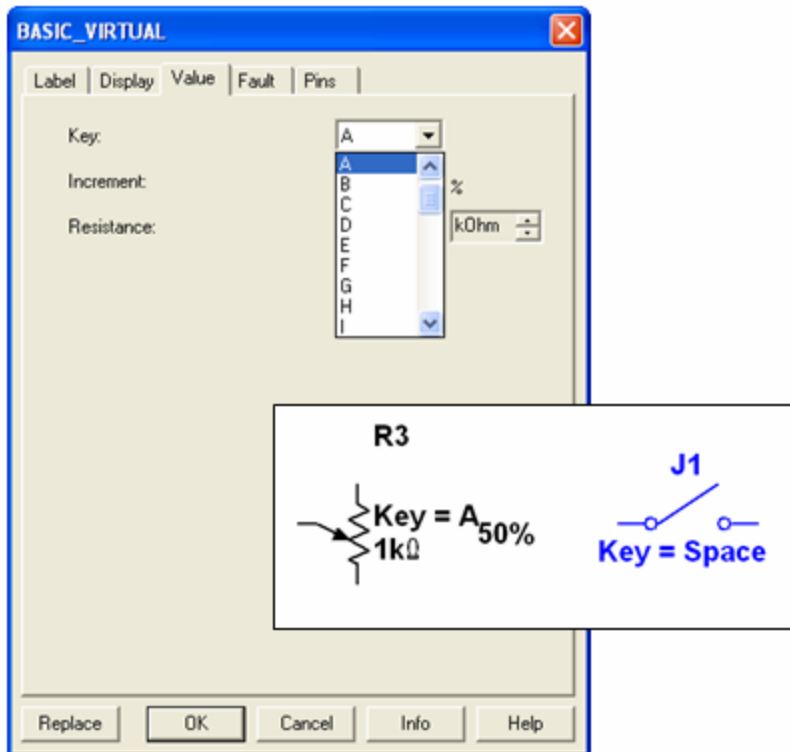
Examples Interactive Components

Component Characteristics

The following fields are visible from the Component Browser.

| Field	Description	Example
Database	Name of Multisim database in which the component is stored.	User
Group	Name of the group to which the component belongs.	TTL
Family	Name of family to which the component belongs.	74S
Component	Name of the individual component.	74S00D
Symbol	Symbol used (either ANSI or DIN) to represent the component during schematic capture.	
Function	Describes the component.	QUAD 2-INPUT NAND
Model Manuf.\ID	Name of the company that manufactures the component and the component's ID.	Texas Instruments\74S00
Footprint Manuf.\ Type	Footprint for the component (real components only) and the package type. Used in Ultiboard or other vendors' PCB layout products.	DO14

Component Information

The Component Browser

The Component Browser is used to select components for placement onto the schematic. To access the Component Browser, click on any icon in the parts bin, or select Place/Component. The default keyboard shortcut to place a component is Ctrl-W.Double-click on the desired component to place it on the schematic. The component will "ghost" the mouse cursor until the left mouse button is clicked again to place the component.

The Parts Bin or Component Toolbar
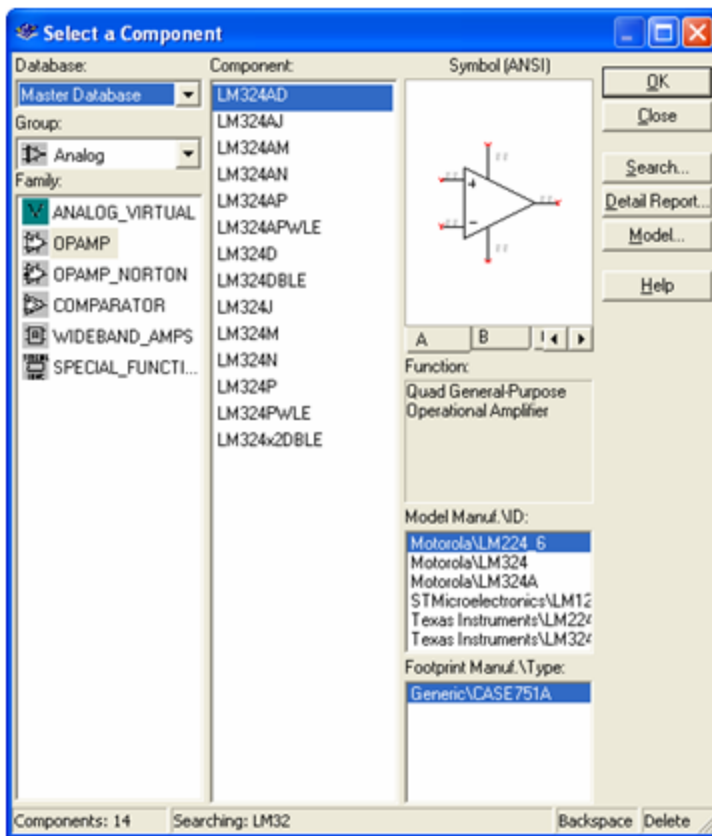
The Component Browser

To search this view, simply start typing the name of the desired component, and the browser will automatically display matching candidate parts. Optionally, for a more detailed search, click on the Search button.

The Component Browser shows the current database in which the displayed parts are stored. Multisim organizes the parts by group, and family. The browser also shows the symbol, a description of the component in the Function field, the model, and the footprint / manufacturer.

The wildcard character '*'can be used to match any set of characters. For example "LM*78" would match components "LM*AD" would return both "LM101AD" and "LM108AD", among others.

Note:Any component may have multiple models associated with it. Each model may account for varying physical characteristics of the component. For example, the LM358M opamp has five visible pins, but only three of them are used in one model, ignoring the power supply terminals. More information about models can be found by selecting the desired model from the Model Manuf.\ID field, and click on the Model button.
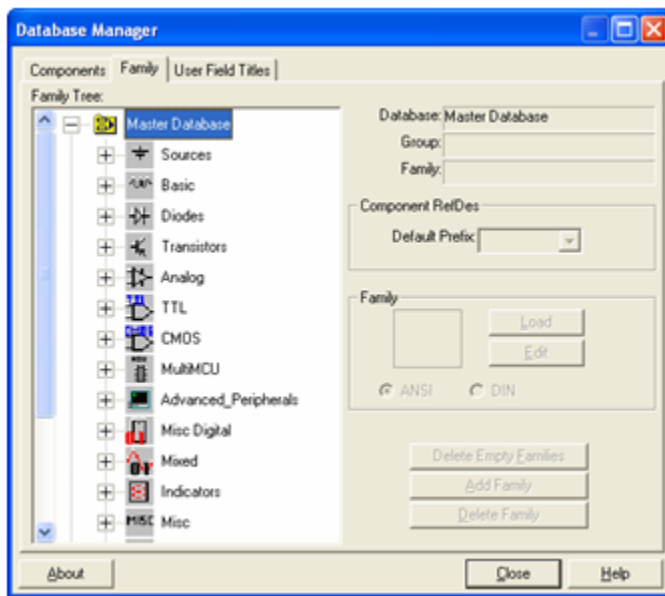
Databases

There are three levels of database provided by Multisim:

- The Master Database is read-only, and contains components supplied by Electronics Workbench.
- The User Database is private to the individual user logged onto the computer. It is used for components built by an individual that are not intended to be shared.
- The Corporate Database is used to store custom components that are intended to be shared across an organization. The Corporate Database can be shared on a network.

Database management tools are supplied in order to move components between databases, merge databases, and edit them. All the databases are divided into groups and then into families within those groups. When a designer chooses a component from the database and drops it onto the circuit, a copy of the component is placed onto the circuit. Any edits made to the component in the circuit do not affect the original database copy.

Edits made to the component in the database do not affect the previously placed components, but will affect all subsequently placed components of that type. When a circuit is saved, component information is saved in the Multisim file. On load, the user has the option to keep the loaded parts as is, to make copies to place into their user or corporate database, or to update similarly-named components with the latest values from the database. Note: The Database Manager can be opened by selecting Tools/Database/Database Manager. To edit Master Database parts, copy them to the User or Corporate Database.
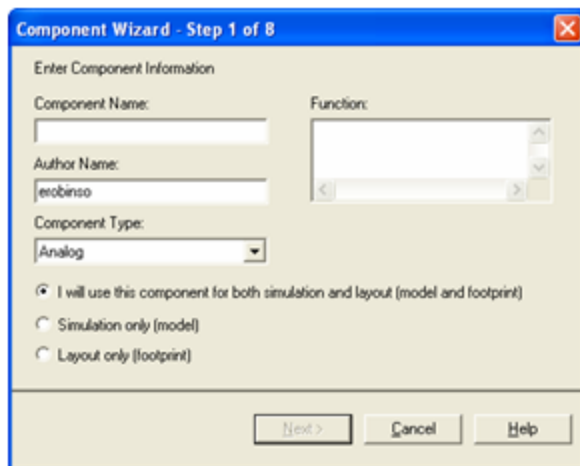

Database Manager

Creating Custom Components

Multisim includes the ability to create and edit components to satisfy the needs of any design. The two methods available are the Component Wizard, and the Component Properties dialog box.

To access the Component Wizard, select Tools/Component Wizard. The component wizard allows designers to enter all pertinent component information, such as symbol, and SPICE model (Figure 7).
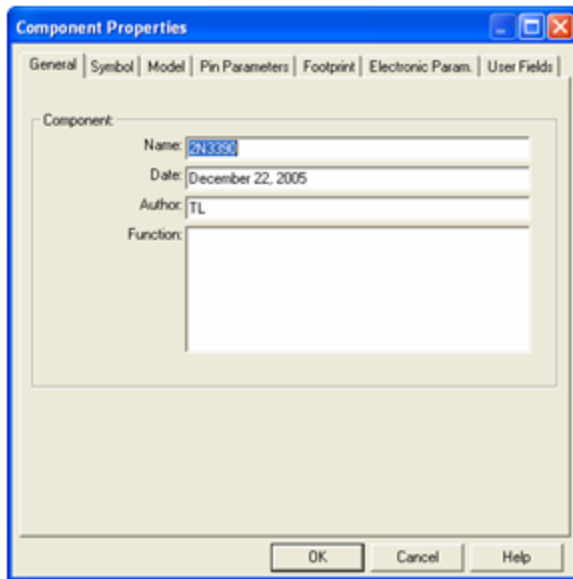
To access the Component Properties dialog box, double-click on a placed component, click on the Value tab, and click the Edit Component in DB button (Figure 8).

Component Wizard

Component Properties Dialog
Box

More details on creating custom components are available in the helpfiles.

Exercise: Finding and Placing Components in National Instruments
Multisim

Approximate time to complete: 10 minutes

This exercise has been designed to introduce users to the component
browser. By the end of this exercise, you should be able to open the
component browser, search and find components that you need, and learn
more information about those components using the various fields of the
browser.

Objectives

- Learn how to use the component browser to locate desired
  components.
- Become familiar with the information that the component browser
  provides.

Procedure

1. Open a new schematic window (File/New/Schematic Capture).
2. Bring up the component browser by clicking on the Sources button of
   the Parts Bin (or Components Toolbar).

1. Locate and place a Groundsymbol onto the schematic. It is found in
   the POWER_SOURCES Family.
2. Use the Searchtool to locate the Analog Devices OP297AZ.

   1. From within the component browser, click Search.
   2. Type "OP297AZ" into the Component field, when you locate the
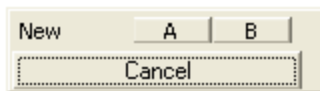   part click OK. Do not place the component yet.

3. How many different varieties of this Opamp are in your database? _____

(Hint:Try including a wildcard '*' in your search: "OP297*")

1. 1. How many sections does this chip have? _____

(Hint: Sections are lettered from A-Z)

1. 1. Should you place the opamp, you will have the option of choosing section A or B. You do not need to place the opamp at this time.

1. 1. Use the Search tool to locate the 74S04D Hex Inverter.

2. When you place the inverter, you are given the option of placing any of the six inverters. Place the A gate. Notice that you now have the option of placing a new chip, or continuing to place gates of the existing U1.
3. Place another "A" gate. What is the default reference designator for this new inverter? _____

Electrical and Electronic Component Manipulation in National Instruments Multisim

Component Placement, Rotation, Selection and Wiring

Placement, Rotation, and Selection

Once parts have been selected from their respective database, it is time to place them onto the schematic, and wire them together. Double-clicking on a component in the browser will attach that component to the cursor. This behavior is known as "ghosting". Ghosting helps guide users when placing components anywhere on the schematic by left-clicking at the desired location.

Components can also be rotated while ghosting, and any time after placement. To rotate a part while ghosting, press Ctrl-R. Ctrl-R will also work when a placed component is selected. Placed components can also be rotated by right-clicking on them and selecting 90 Clockwise or 90 CounterCW.
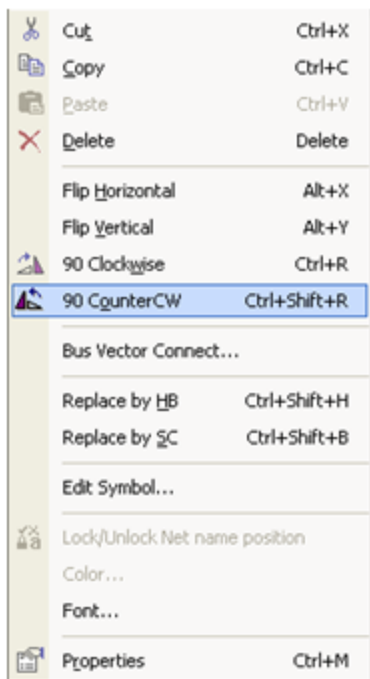
Component
Rotation
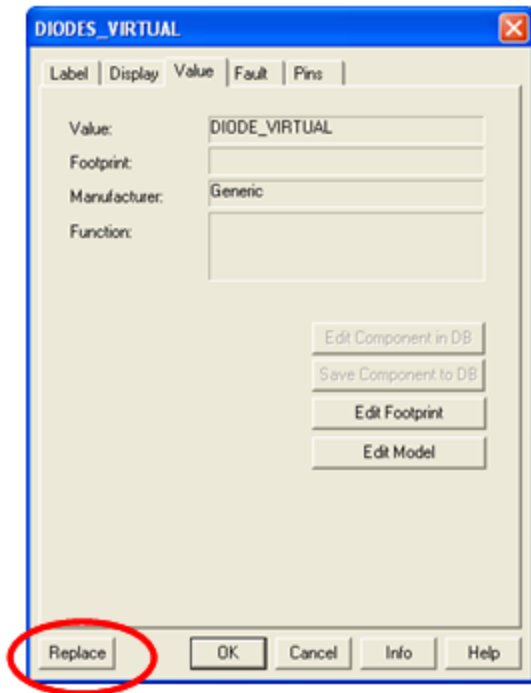
Replacing Components

To select a component, simply-left click on it. To select multiple components, click and drag to create a selection around the desired components. A dashed line indicates that a component is selected. Individual elements of a symbol can also be selected, such as the component value and reference designator. To select these, left-click on the desired text or graphic element.

Holding the Shift key while selecting allows multiple components to be selected or de-selected.

Components can be replaced by right-clicking on them then selecting Replace Component(s) from the right-click menu. Users can then select the replacement components from the newly opened component browser. Multisim will connect the new component to the same nets as the original component.

Wiring

Multisim provides modeless operation — the action performed by the mouse cursor is dependent on the cursor's location. There is no need to select a tool, or mode when working with Multisim. The cursor will change depending on what object is underneath it. describes the different icons that the mouse cursor will display.
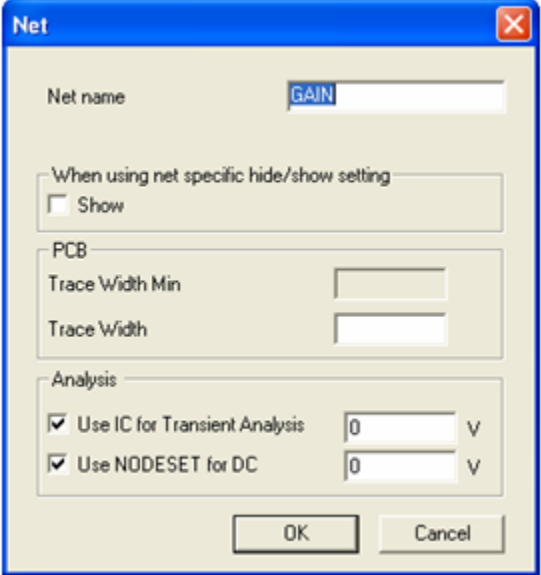
When the cursor is over a pin or terminal of a component, that component can easily be wired by left-clicking. When the cursor is over an existing wire and near a pin or terminal the net can easily be re-wired.

Left-click on the terminal to begin wiring, and to finish the wire, left-click on the destination terminal.

When a wire is placed, Multisim will automatically assign it a net number. Net numbers increase sequentially, beginning with 1. Ground nets are always numbered 0 — a requirement of the underlying SPICE simulator. To change a net number, or assign it a logical name instead, simply double-click on the wire.

Cursor	Mode
▶	Place or Move Part Select Menu Item or Icon
◆	Place Wire
✕═══	Rewire / Move Wires

Modeless Mouse Cursors

Naming Nets

Autowiring by Touching Pins

Multisim also allows auto-connection of pins to wires, and pins to pins. To automatically connect a component to existing nets or pins, simply place that component so that its pins are touching an existing net or pin.

Step 1

Existing components.

Step 2

Add a component to the workspace.

Step 3

Move the component into contact with the wire and a junction is automatically placed when the mouse is released.

Step 4 (Optional)

Drag the component to a new location....

Step 5

....and note that the component stays attached to the wire.
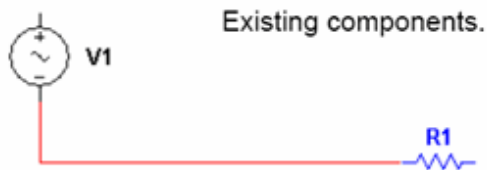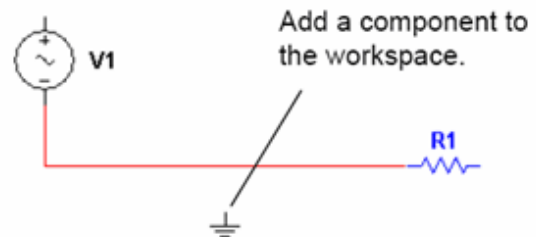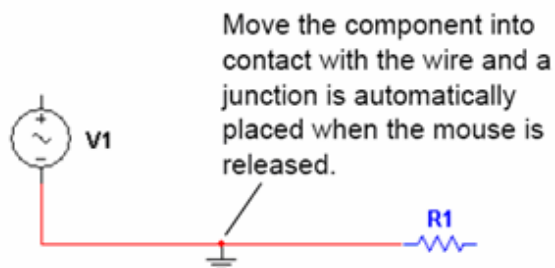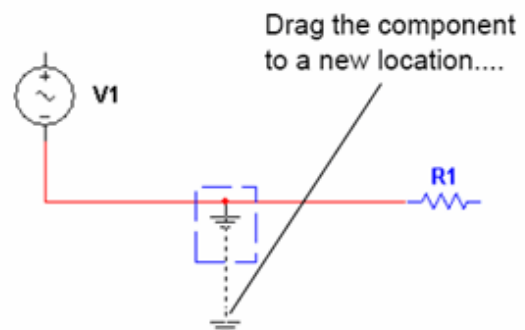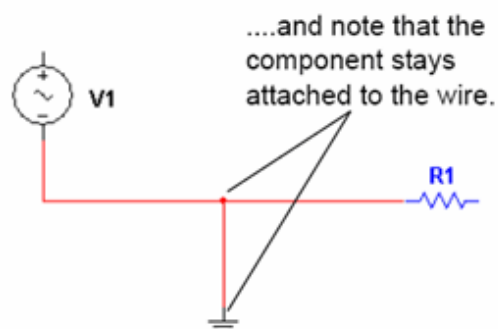
Autowiring by Touching Pins

Autoconnect Passives

Multisim provides the ability to place a component in-line with an existing wire or set of wires. To automatically split an existing wire around a component, simply place that component in-line with the wire (Figure 6).
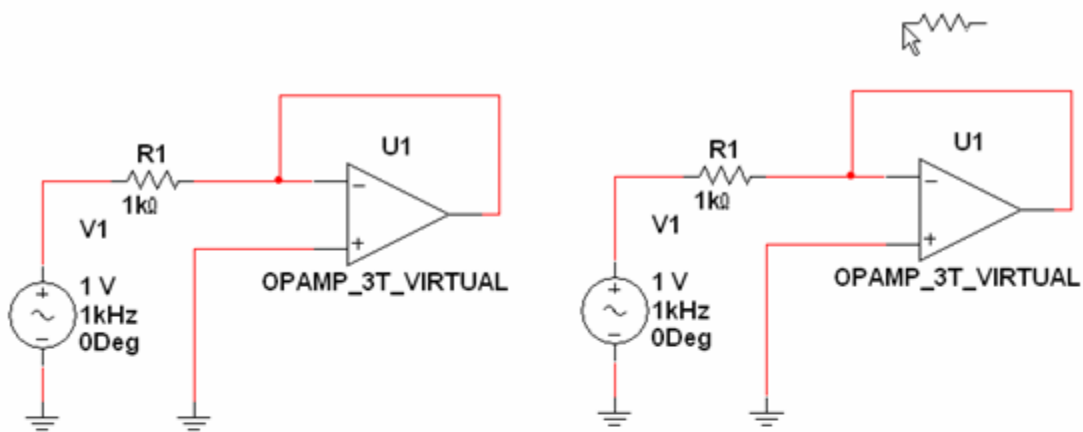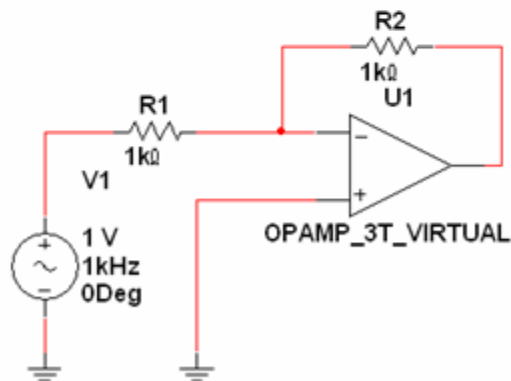
Autoconnect Passives

Exercise: Drawing a Schematic in MultiSim

Approximate time to complete: 20 minutes.

This exercise provides a general introduction to Multisim's schematic capture. You will build and wire a basic circuit in Multisim using a variety of means to access parts, experiment with the wiring and run a basic simulation.

Objectives

- Understand the difference between real, virtual, ideal, and interactive parts.
- Build and wire a basic circuit (including virtual wiring).
- Become familiar with and set wiring options.

Procedure

1. Build your own version of circuit 40kFILTER1_Complete.ms9 as pictured below. Select the required components from the Master Database (Place/Component) and the In-Use List. Set component values as identified below. Note: Components R1, R2 and C2 are all virtual parts.

Place a capacitor between points A and B.

A   B

R2
191kΩ

Input   R1   C2   V2
40.2kΩ   270pF   15 V   U19A

R3 1KΩ_LIN
Key = A   50%

V1
40kHz
0.3 V

Output

LM258N
V3
15 V
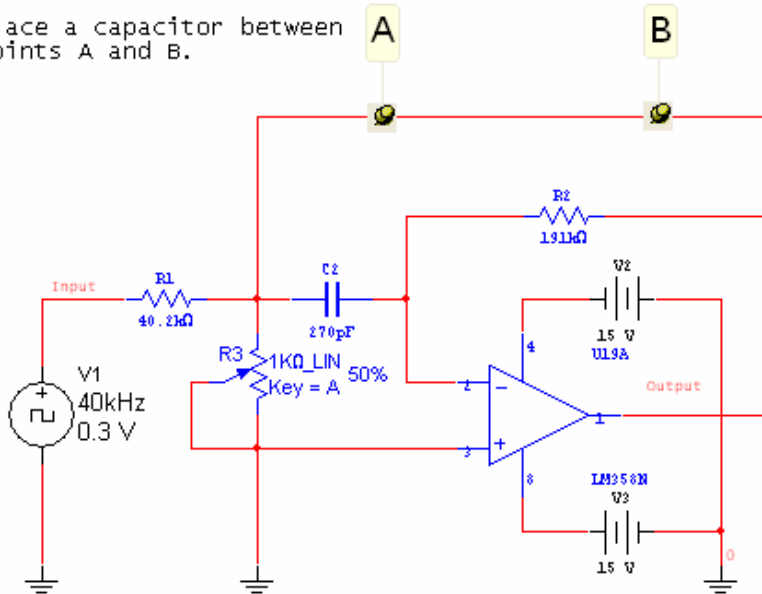
Notes:

All components except for the opamp are **Virtual Components**.

The opamp has been flipped vertically

Bandpass Filter

1. To wire your circuit, point at a component terminal so that you see the cursor changes to crosshairs and left-click. Move the pointer (while dragging the wire) to the second component terminal and left-click to terminate.
2. Using the Replace function by right clicking on the R2, choose Replace Component(s) and substitute the virtual resistor (R2) with a real resistor (Basic/Resistor) of your choice.
3. Double-click on the virtual components to see how they can be used to set variable parameters.
4. Rotate and move a component within the circuit to see how movement of components affects the wiring. Components can also be rotated while being placed from the database.
5. Select the virtual capacitor from the In-Use List and place it between Points A and B in the circuit. Notice the how it is automatically connected, and has a capacitance of 270 pF.

SOLUTION

Creating Circuits in Multisim

Circuit Wizards

Multisim provides several circuit wizards, which can aid designers by quickly producing circuits to match specifications. The circuits wizards provided are listed in . To use a circuit wizard, select Tools/Circuit Wizards.

Circuit Wizards
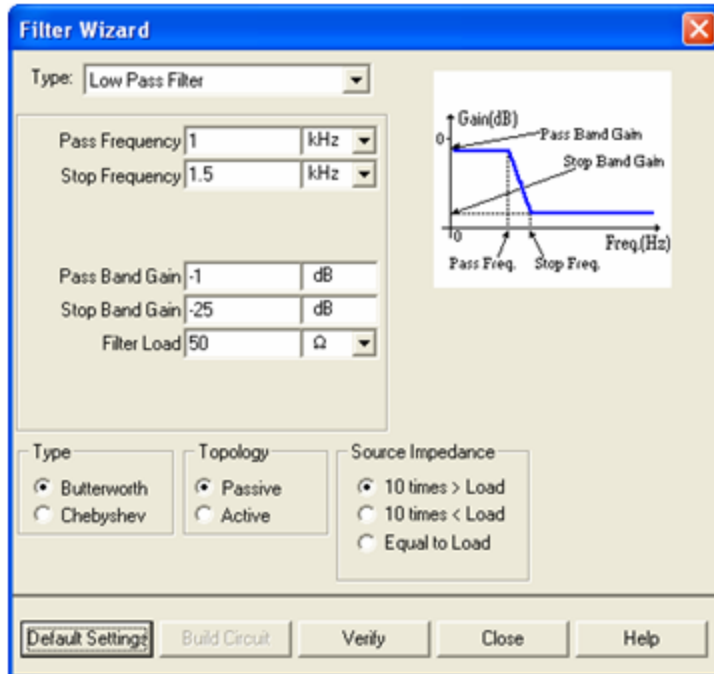
Filter Wizard Dialog Box

Use the 555 Timer Wizard to build astable and monostable oscillator circuits that use the 555 timer.

The Multisim Filter Wizard helps design numerous types of filters by entering the specifications into its fields.

The Common Emitter BJT Amplifier Wizard helps design common emitter amplifier circuits by entering the desired specifications into its fields. The Multisim MOSFET Amplifier Wizard helps design MOSFET amplifier circuits. The Multisim Opamp Wizard helps design the following opamp circuits. Users can enter the desired specifications in its fields:

- Inverting Amplifier.
- Non-inverting Amplifier.
- Difference Amplifier.
- Inverted Summing Amplifier.
- Non-inverted Summing Amplifier.

- Scaling Adder.

Using Electrical Rules Checking in National Instruments Multisim

Electrical Rules Check (ERC)

The Electrical Rules Check creates and displays a report detailing connection errors (such as an output pin connected to a power pin) and unconnected pins. Once the circuit is wired, check the connections for correctness based on the rules set up in the Electrical Rules Check dialog box.

Depending on your circuit, you may wish to have warnings issued if some types of connections are present, error messages for other connection types, and no warnings or errors for other connections. You control the type of connections that are reported when ERC is done by setting up the rules in the grid found in the ERC Rules tab of the Electrical Rules Check dialog box.

ERC may be run over an entire design, or only across certain areas of a design. When an ERC is run, any anomalies are reported into a results pane at the bottom of the screen and the circuit is annotated with circular error markers. Clicking on an error will center and zoom on the error location.

The ERC Options tab and ERC Rules tab are used to configure the ERC.

To run the electrical rules check:

1.Select Tools/Electrical Rules Check to display the Electrical Rules Check dialog box.

2.Set up the reporting options using the ERC Options tab (Figure 1).

3.Set up the rules using the ERC Options tab (Figure 2).

4.Click OK. The results display in the format selected in the Output box in the ERC Options tab.
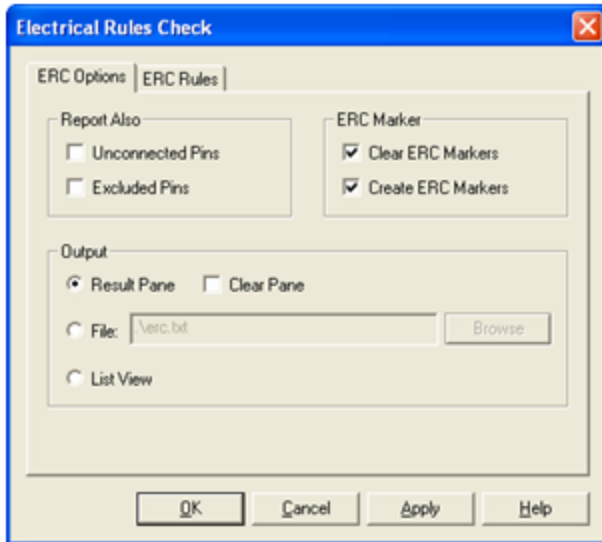
ERC Options Tab
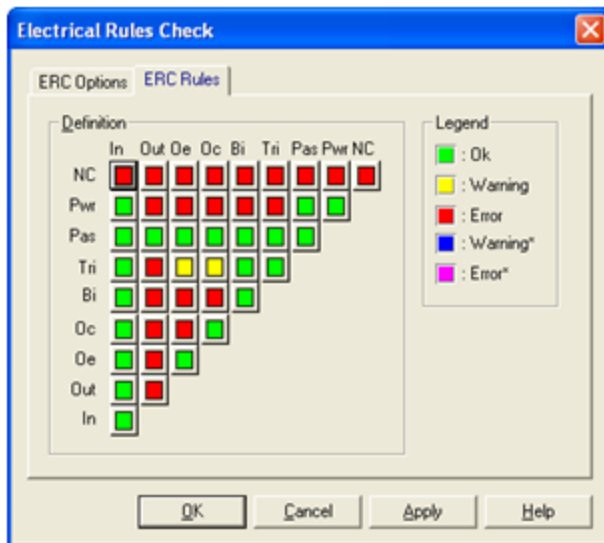
ERC Rules Tab

Creating and Using Sub-Circuits in National Instruments Multisim

Sub-circuits and Hierarchical Blocks

Multisim provides the ability to handle increasingly complex designs. In addition to multi-sheet designs, users can create sub-circuits (SC), and hierarchical blocks (HB) to modularize repetitive circuits, or to abstract sophisticated designs.

Subcircuits are useful for compacting existing designs that would be best kept in a single file. Hierarchical blocks are better suited for design reuse because they are stored in separate files and can be accessed for other designs.

Hierarchical blocks and subcircuits are functionally identical; the only difference is in how their contents are stored on disk.

Hierarchical blocks and subcircuits can be created using two methods, the first method is to highlight an existing section of a circuit, and select Place/Connectors/HB/SC Connector. The second method is described below.

To place a new hierarchical block (2nd method):

1. Select Place/New Hierarchical Block and enter a filename.
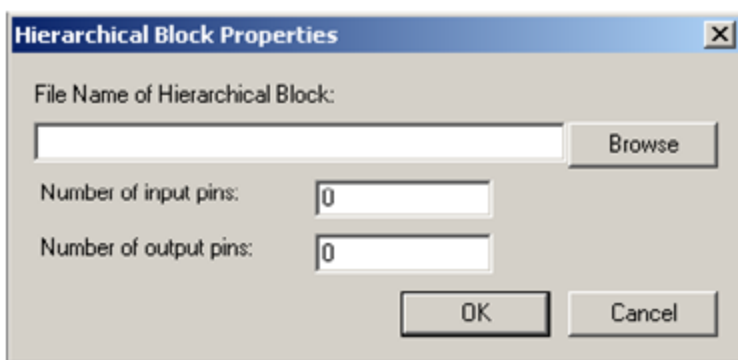
Hierarchical Block Properties Dialog Box

or

1. Click on Browse, navigate to the folder where you would like to save the hierarchical block, enter a name and click Save. You are returned to the Hierarchical Block Properties dialog box.
2. Enter the number of pins desired and click OK. A "ghost" image of the new hierarchical block appears. Click where you want the hierarchical block to appear.
3. Double-click on the new hierarchical block and select Edit HB/SC from the Hierarchical Block/Subcircuit dialog box that displays. A circuit window that contains only the entered pins displays.
4. Place and wire components as desired in the new hierarchical block.
5. Wire the hierarchical block into the circuit.
6. Save the circuit.

Note: If you move or re-name a hierarchical block relative to the main circuit, Multisim will not be able to find it. A dialog box will ask you to provide the new location for the hierarchical block.

To place an existing hierarchical block from a file, select Place/Hierarchical Block from file and follow the same procedure.

To place a new subcircuit:

1. Select Place/New Subcircuit. The Subcircuit Name dialog box appears.
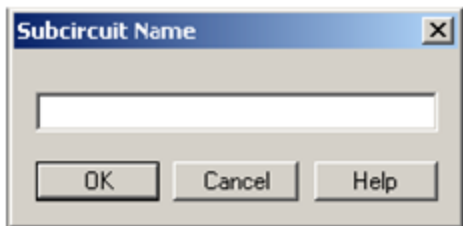
Subcircuit Name Dialog
Box

1. Enter the name you wish to use for the subcircuit, for example, "PowerSupply" and click OK. Your cursor changes to a "ghost" image of the subcircuit indicating that the subcircuit is ready to be placed.
2. Click on the location in the circuit where you want the subcircuit placed (you can move it later, if necessary). The subcircuit appears in the desired location on the circuit window as an icon with the subcircuit name inside it.
3. Double-click on the new subcircuit and select Edit HB/SC from the Hierarchical Block/Subcircuit dialog box that displays. An empty circuit window appears.
4. Place and wire components as desired in the new hierarchical block.
5. Select Place/Connectors/HB/SC Connector, and place and wire the connector as desired. Repeat for any other required Connectors. When you return to the main circuit, the symbol for the subcircuit will include pins for the number of connectors that you added.
6. Wire the subcircuit into the circuit.

Replacing Components with Hierarchical Blocks or Subcircuits

Multisim allows users to easily replace existing components with a hierarchical block or subcircuit. Simply select the components which comprise the desired subcircuit or hierarchical block, and select Place/Replace by Hierarchical Blockor Place/Replace by Subcircuit.

Spreadsheet View

The Spreadsheet View provides a global perspective on object properties. It allows fast advanced viewing and editing of parameters including component details such as footprints, Reference Designators, attributes and design constraints.

The Spreadsheet View can also be used to modify groups of components at a time. The view can be sorted by any column in either ascending or descending order. You can also export the contents to Microsoft Excel® for further reports.

Generating Schematic Reports and Annotation in National Instruments Multisim

Schematic Reports and Annotation

Multisim allows you to generate a number of reports: Bill of Materials (BOM), Component Detail Report, Netlist Report, Schematic Statistics, Spare Gates and the Cross Reference Report. A BOM lists the components used in your design and therefore provides a summary of the components needed to manufacture the circuit board. Information provided in the Bills of Materials include:

- quantity of each component needed.
- description, including the type of part (example: resistor) and value (example: 5.1 kohm).
- Reference Designator of each component.
- package or footprint of each component.

The Component Detail Report shows all information stored in the Multisim database for a particular component.

The Netlist Report provides the following circuit connectivity information for each component:

- Net name.
- Page (filename).
- Pin (logical pin name).

The Cross Reference Report provides a detailed list of all components and their locations in the design.

The Schematic Statistics Report provides the quantity of the following in a circuit:

- Components — total number of components. (Real components + virtual components).
- Real components — components that can be purchased.
- Virtual components — components that cannot be purchased.

- Gates — total number of gates used in the design.
- Nets — total number of connections between pins.
- Pins in nets.
- Unconnected pins.
- Total pins — pins in nets and unconnected pins.
- Pages.
- Hierarchical blocks — total number of hierarchical blocks, unique or otherwise. Instances of hierarchical blocks equals the number of copies of the same hierarchical block.
- Unique hierarchical blocks — total number of unique hierarchical blocks.
- Subcircuits — total number of subcircuits, unique or otherwise.
- Unique subcircuits.

The Spare Gates Report lists unused gates on multi-section components. Note: Multisim provides a gate optimizer which will automatically and efficiently pack multisection parts onto the minimum number of chips. To run the gate optimizer select Tools/Rename/Renumber Components and then select Gate Optimizer.

Graphical Annotation

Multisim provides an easy method for graphically annotating your circuits. The Graphic Annotation toolbar allows placement of the following items: text, lines, multilines, rectangles, ellipses, arcs, polygons, pictures, and comments. To place graphical elements without using the toolbar, right-click on the schematic and select Place Graphic.

Graphical Annotation
Toolbar

Circuit Description Box

In addition to adding text to a particular portion of a circuit, you can add general descriptions to your circuit using the Circuit Description Box. You can also place bitmaps, sound and video in the Circuit Description Box.

The contents of the Circuit Description Box are viewed in the top pane of the Circuit Description Box window (select View/Circuit Description Box). To edit the contents of the Circuit Description Box, select Tools/Description Box Editor.

Title Blocks

A powerful title block editor allows you to create customized title blocks. If desired, a title block can be included on every page of your design.

Various fields in the title block are automatically filled in depending upon the context and various document properties. When designing the title block, you choose one of the pre-defined fields or a custom field. You choose appropriate fonts depending upon your language of preference. To edit an existing, or to create a new title block, select Tools/Title Block Editor.

Title blocks can include elements such as text, lines, arcs, Bezier curves, rectangles, ovals, arcs, bitmaps, and so on.

To place a title block, select Place/Title Block. The title block can be automatically placed in any corner by right-clicking on the title block and selecting Move To. To populate fields of each title block, simply double-click the title block.

Schematic Transfer to Ultiboard and Other Packages

Multisim includes a single-click feature which will transfer the current design to an installed copy of Ultiboard. Select Transfer/Transfer to Ultiboardto initiate the PCB design process. Designs can be easily forward and back-annotated using the same Transfer menu.

In addition to a transfer mechanism between Multisim and Ultiboard, designers also have the option to transfer designs to one of several other PCB layout packages. Note: Transferring designs to 3rd party layout packages may require matching Multisim components to components in the database of the chosen PCB layout program.

Simulating Circuits using the SPICE in National Instruments Multisim

Simulation Overview

While a good design naturally follows from quality schematics, truly great designs can made with the help of simulation. Multisim provides powerful simulation capabilities and features which are simply unavailable in other EDA packages.

Simulating a design can result in fewer design iterations and less chance of errors in the prototype stage of product development. When a design is simulated at the front end of the design process, the number of design cycles can be significantly reduced.

In addition to a world-class SPICE simulator, Multisim also includes XSPICE simulation, enabling fast simulation of digital other components.

Patented co-simulation techniques allow designers to simulate VHDL modeled components along with the rest of a circuit. With MultiMCU, certain microcontrollers can be simulated in the very same mixed-mode environment. MultiMCU is not available for all versions of Multisim.

Using the Interactive Simulator

To begin a simulation, ensure that the circuit has all the necessary prerequisites. All circuits must include a ground reference, and a source. Once the circuit is ready for simulation, click the Run / Stop Simulation button

Run/Stop
Simulatio
n Button

or press F5. An interactive circuit simulation will begin.

The settings that are used for interactive simulation can be viewed and modified by selecting Simulate/Interactive Simulation Settings.Figure 1 below illustrates some of the settings available for interactive simulation. The default end time for the simulation is 1e+30 seconds, or around 3.17e+30 billion years. By default time steps will be generated automatically.
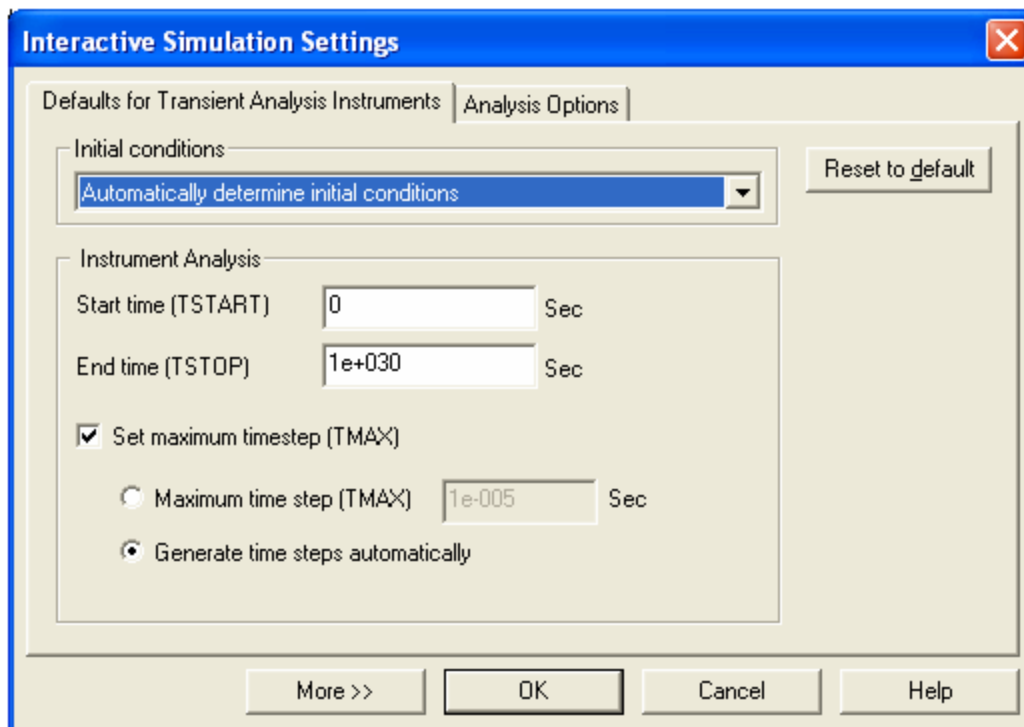
Interactive Simulation Settings

To view results of this simulation, the measurement probe

The
Measuremen
t Probe

can be used dynamically. Simply click on the probe icon and the mouse cursor will turn into a probe. The mouse can then be moved over any net to view the following metrics:

- Voltage (instantaneous, peak-to-peak, RMS, and DC).
- Frequency.

Virtual instruments can also display simulation results. Virtual instruments are covered later in this section.

Multisim also provides more traditional SPICE analyses, which are run through the Grapher/Analyses List toolbar button,

Grapher/Analyse
s List toolbar
button

or by selecting Simulate/Analyses. Analyses are described in greater detail later in this section.

Handling Simulation Errors

Sooner or later, even the most experienced SPICE users will run into a SPICE simulation error. Multisim includes a simulation advisor to help discover and fix the source of troubling errors.

When an error message appears such as the one in Figure 5, click the Adviser button to access the available help.
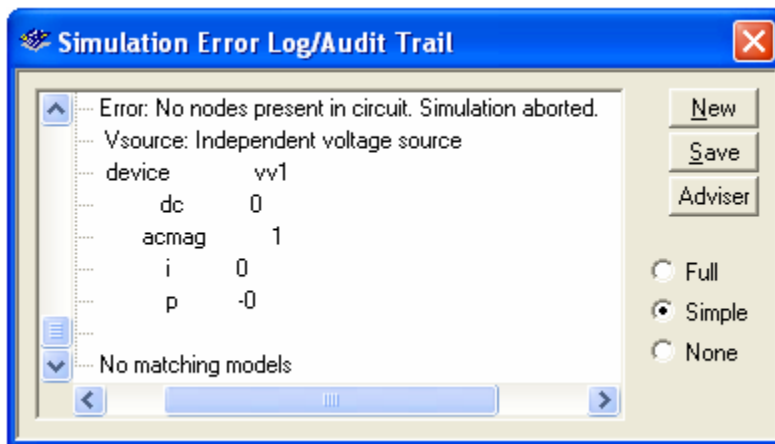
Simulation Error Log Dialog Box
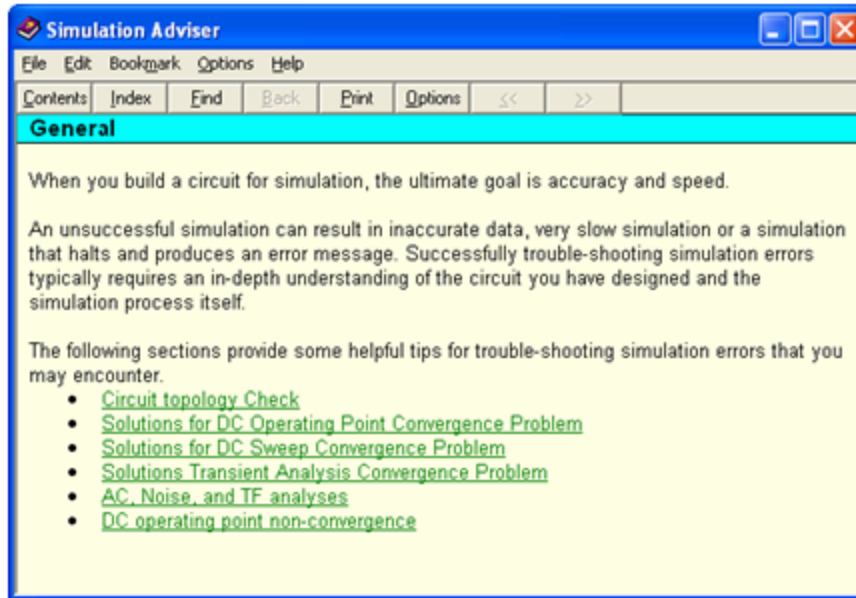
Simulation Advisor

Two of the most often encountered errors are timestep errors, and singular matrix errors. below provides the most common solutions to these simulation errors.

Action	Setting	Default	New Value
Error: Time Step Too Small			
Change initial conditions	Initial Conditions	Automatically Determine	Set to zero
Increase maximum time step	TMAX	1×10^{-5}	1×10^{-3}
Increase relative error tolerance	RELTOL	0.001	0.01
Decrease shunt resistance	RSHUNT	1×10^{12}	1×10^{9}
Change Integration Method (High power or circuits with switches)	METHOD	Trapezoidal	Gear
Error: Singular Matrix Error			
Check your wiring			
Decrease diagonal element threshold	PIVTOL	1×10^{-13}	Decrease to smaller value than listed in error message
Increase allowed number of fill ins	PIVREL	1×10^{-13}	1×10^{-11}

Common Solutions to Simulation Errors

Instrumenting a Circuit Simulation in National Instruments Multisim

Instrumenting a Simulation in MultiSim

Virtual instruments are components in Multisim which model real-world bench-top instruments. Examples of virtual instruments in Multisim include Oscilloscopes, Function Generators, Network Analyzers, and Bode Plotters.

Virtual instruments provide designers with an easy and intuitive method for interacting with their circuits as they would in a testing or prototyping phase.

In addition to the provided virtual instruments, designers with familiar with National Instruments LabVIEW can create their own custom instruments from scratch. For example, one could create a custom noise generator to model electromagnetic interference.

Custom virtual instruments written in NI LabVIEW can also acquire real-world data and use the data to drive simulations, and can send generated data to analog output hardware, allowing simulated data to control real-world devices. The LabVIEW development suite is required to create custom LabVIEW instruments, but not necessary to run existing LabVIEW based instruments.

To place a virtual instrument, select the desired instrument from the Instruments toolbar (Figure 1). To view the front panel of the instrument, double-click on the instrument icon. Make connections to the terminals of the instrument icon as you would to any other component.

Multisim also provides designers with simulated benchtop instruments. These are instruments such as the Tektronix TDS 2024 Oscilloscope. They will look and operate exactly according to the manufacturers' user manuals.

Instruments Toolbar

A single circuit can have multiple instruments attached to it, including (for most tiers) multiple instances of the same instrument. In addition, each circuit window can have its own set of instruments. Each instance of any instrument is configured and connected independently.

An overview of the more common instruments is included in this section. For more detailed information on how to use each instrument, consult the Multisim User Guide or helpfile.

Multimeter

Use the multimeter to measure AC or DC voltage or current, and resistance or decibel loss between two nodes in a circuit. The multimeter is auto-ranging, so a measurement range does not need to be specified. Its internal resistance and current are preset to near-ideal values, which can be changed.
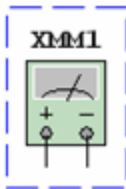
Multimeter
Schematic
Symbol

Multimeter
Front Panel

Function Generator

The function generator is a voltage source that supplies sine, triangular or square waves. It provides a convenient and realistic way to supply stimulus signals to a circuit. The waveform can be changed and its frequency, amplitude, duty cycle and DC offset can be controlled. The function generator's frequency range is great enough to produce conventional AC as well as audio- and radio-frequency signals.

The function generator has three terminals through which waveforms can be applied to a circuit. The common center terminal provides a reference level for the signal.

Function
Generator
Schematic Symbol
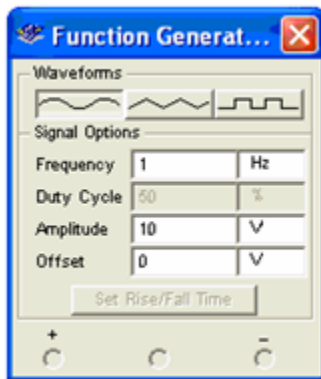
Function Generator Front Panel

Oscilloscopes

Multisim includes several varieties of oscilloscopes. All of the scopes can be controlled like real-world oscilloscopes. Horizontal timing and vertical voltages parameters can be adjusted. Triggering levels, and types are also selectable. Data from the Multisim-exclusive oscilloscopes is available in the Grapher after the interactive simulation has completed by selecting View/Grapher.

Multisim provides the following oscilloscopes:

- 2-Channel.
- 4-Channel.
- Agilent 54622D Mixed Signal Oscilloscope.
- Tektronix TDS 2024 Four Channel Digital Storage Oscilloscope.
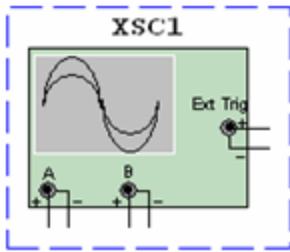
Oscilloscope
Schematic
Symbol
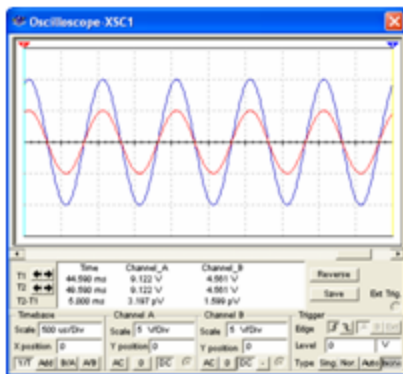
Oscilloscope Front
Panel
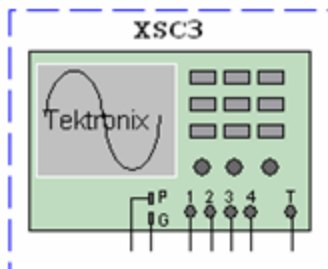
Tektronix
Oscilloscope
Schematic
Diagram

Tektronix Oscilloscope Front Panel

Bode Plotter

The bode plotter displays the relative phase or magnitude response of an input and output signal. This is especially useful when characterizing the bandwidth of filter circuits.

Function
Generator
Schemati
c Symbol
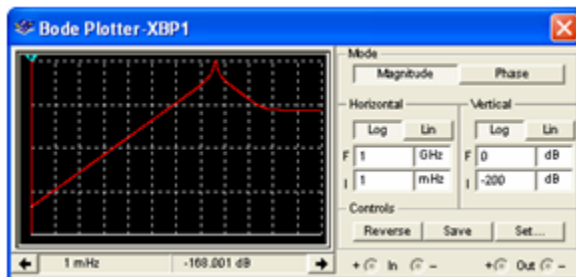
Bode Plotter Front Panel

Spectrum Analyzer

The spectrum analyzer is used to measure amplitude versus frequency. This instrument is capable of measuring a signal's power and frequency components, and helps determine the existence of harmonics in the signal.

The spectrum analyzer displays its measurements in the frequency domain rather than the time domain. Usually the reference frame in s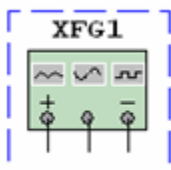ignal analysis is time. In that case, an oscilloscope is used to show the instantaneous value as a function of time. Sometimes a sine waveform is expected but the signal, rather than being a pure sinusoidal, has a harmonic on it. As a result, it is not possible to measure the waveform's level. If the same signal is displayed on a spectrum analyzer, its amplitude is displayed, along with its frequency components, that is, its fundamental frequency and any harmonics it may contain.
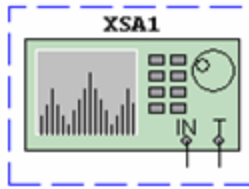
Spectrum
Analyzer
Schematic
Symbol
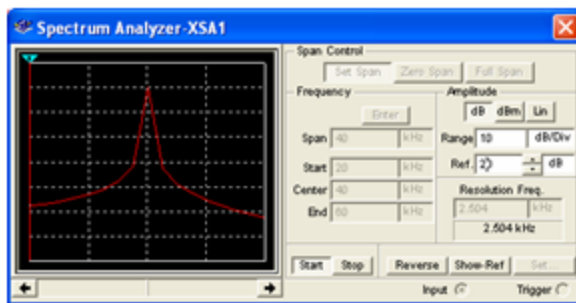
Spectrum Analyzer Front Panel

Exercise: Instrumenting Simulated Circuits in National Instruments Multisim

Approximate time to complete: 20 minutes

This exercise will demonstrate the interactive simulator, and virtual instruments. By the end of this exercise, users will be familiar with placing instruments, opening their front panels, and configuring various options.

Objectives

- Learn how to place and connect virtual instruments.
- Gain experience configuring instruments.

Procedure

1. Load circuit [40kFilter2.ms9](). Refer to while performing steps 2 - 4.
2. Replace the Clock Source with a Function Generator. Once placed, double-click to open the instrument panel and adjust its settings as follows:

- Waveform = sinewave
- Amplitude = 1 V
- Frequency = 40 kHz

1. Close the instrument panel.
2. Attach the Bode plotter between the input and output nodes. Double-click to open the instrument and adjust its settings as follows, then simulate and observe the output:

- Set Magnitude
- Horizontal I (Initial) = 1 kHz, F (Final) = 1 MHz
- Vertical I (Initial) = -50 dB, F (Final) = 10 dB

1. Attach an oscilloscope to monitor the input and output voltages.Double-click on the oscilloscope icon to adjust its settings as follows:

- Set Timebase = 20 us/Div
- Channel A= 1 V/Div
- Channel B = 1 V/Div

1. The color of the wire segment connected to an instrument controls the color of the trace on the instrument display. Make sure that the wires from the output of the op-amp are blue. If they are not, then right-click on a wire segment and choose "Wire Color" to change the setting. Now, simulate and observe the output.
2. Change the value of the Potentiometer (R3) by pressing "A" to increase the resistance and "Shift-A" to decrease the resistance during simulation. Observe the response on the oscilloscope. Note: The Bode Plot response will be updated only after you re-simulate the circuit.

1. While the simulation is still running, use the "Measurement Probe" to view the voltage levels in the circuit. It is found at the end of the instrument panel.

 SOLUTION

Creating LabVIEW Instruments to Instrument Circuit Simulations in National Instruments Multisim

Creating LabVIEW Instruments in MultiSim

Multisim ships with four different virtual NI LabVIEW instruments: a microphone, speaker, signal generator, and signal analyzer. The symbols are shown in Figure 1 and the speaker front panel is shown in Figure 2.

Microphone - Records audio from the computer sound device and outputs the sound data as a signal source.

Speaker - Plays input signal using the computer sound device.

Signal Generator - Generates and outputs a sine, triangle, square, or sawtooth wave.

Signal Analyzer - Displays time-domain data, auto power spectrum, or running average of input signal.
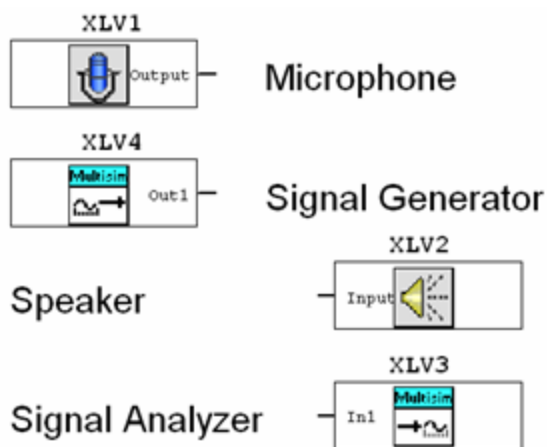
NI LabVIEW Instrument Symbols

Sample NI
LabVIEW Front
Panel

Custom instruments can be created using the NI LabVIEW graphical development environment. Instruments that are created using NI LabVIEW can take advantage of the full functionality of the NI LabVIEW development system including data acquisition, instrument control, mathematical analysis, and so on.

For example, you can create an instrument that acquires data from the real world using a National Instruments data acquisition device or modular instrument. Multisim will then use that data as a signal source for circuit simulation.

You can also create an instrument that displays simulation data simultaneously with multiple measurements (running average and power spectrum, for example) made from that simulation data.

NI LabVIEW instruments can be either input instruments or output instruments. Input instruments receive simulation data for display or processing. Output instruments generate data to use as a signal source in

simulation. A NI LabVIEW instrument cannot be both an input and output instrument.

Additionally, input and output instruments behave differently. Input instruments continuously receive simulation data from Multisim while simulation is active.

In contrast, at the beginning of circuit simulation, output instruments will first generate a finite amount of data and return that data to Multisim. Multisim will then use that data for circuit simulation. Output instruments do not continuously generate data while simulation is running. To cause an output instrument to generate new data, you must stop and restart simulation.

Output instruments allow the user or creator of the instrument to determine whether the instrument repeats the output data. The LabVIEW instrument must be configured to repeat its output data periodically, or else the simulator will assume the output is 0V once it has no data remaining from the instrument. Alternatively, if you configure the instrument and repeat the output data, the instrument will loop and repeat its output periodically until the simulation has stopped.

Input instruments allow the user or creator of the instrument to set a sampling rate. This sampling rate is the rate at which the instrument receives data from Multisim. This sampling rate is analogous to the sampling rate you would set for a physical data acquisition device or modular instrument that acquires data from the real world. You should observe the Nyquist sampling theorem when choosing a sampling rate for your instrument. Note that the higher the value of the sampling rate, the slower simulation will run.

To create and modify NI LabVIEW instruments, you must have the NI LabVIEW 8.0 (or later) Development System.

To use NI LabVIEW instruments, you must have the NI LabVIEW Run-Time Engine installed on your computer. The version of this Run-Time Engine must correspond to the version of the NI LabVIEW Development System used to create the instrument. The Multisim installer includes the
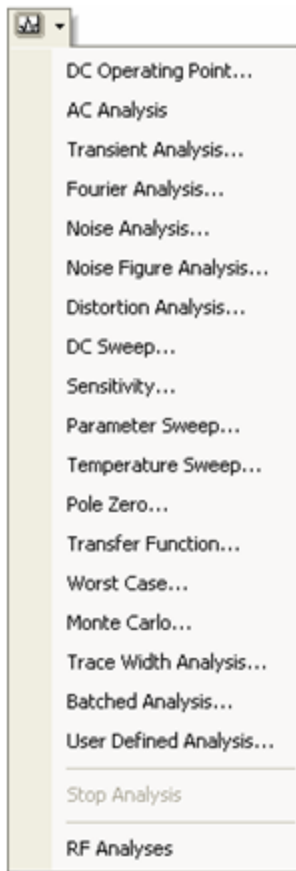
NI LabVIEW Run-Time Engine 8.0 as part of the Electronics Workbench Shared Components installation.
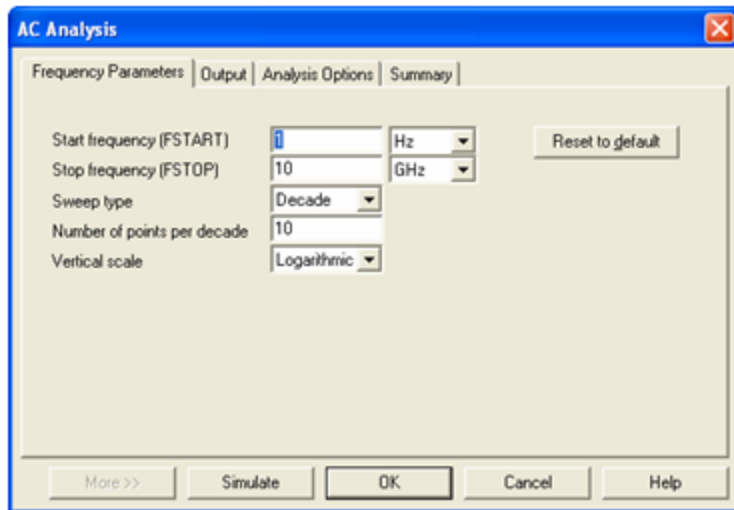
Analyses of Circuit Simulations

Analysis of Circuit Simulation

Multisim offers many analyses, all of which utilize simulation to generate the data for the desired analysis. These analyses range from quite basic to extremely sophisticated, and often require one analysis to be performed as part of another. To configure and begin an analysis, select Simulate/Analyses, and choose the desired analysis. Figure 1 lists all available Multisim analyses. For each analysis, you configure the settings that tell Multisim how to exactly perform the desired analysis. In addition to the analyses provided by Multisim, user-defined analyses can be created based on user-entered SPICE commands.

To prepare an analysis, configure the analysis-specific parameters, such as frequency range for an AC analysis. Output traces must also be selected here. It is especially important to name nets appropriately to avoid confusion when analyzing results. The results are displayed on a plot in Multisim's Grapher and saved for use in the Postprocessor. Some results are also written to an audit trail, which can be viewed.

DC Operating Point…
AC Analysis
Transient Analysis…
Fourier Analysis…
Noise Analysis…
Noise Figure Analysis…
Distortion Analysis…
DC Sweep…
Sensitivity…
Parameter Sweep…
Temperature Sweep…
Pole Zero…
Transfer Function…
Worst Case…
Monte Carlo…
Trace Width Analysis…
Batched Analysis…
User Defined Analysis…
Stop Analysis
RF Analyses

Available Analyses

AC Analysis Options Dialog Box

Grapher

The Grapher is the primary tool used to view the results of simulations. Users can view the Grapher by clicking View/Grapher. Additionally, the Grapher opens automatically displayed upon when running an analysis. Elements of the Grapher window are detailed in Figure 3.

The display shows both graphs and charts. In a graph, data are displayed as one or more traces along vertical and horizontal axes. In a chart, text data are displayed in rows and columns. The window is made up of several tabbed pages, depending on how many analyses, etc. have been run.

Each page has two possible active areas which will be indicated by a red arrow: the entire page indicated with the arrow in the left margin near the page name or the chart/graph indicated with the arrow in the left margin near the active chart/graph. Some functions, such as cut/copy/paste, affect only the active area, so be sure the desired area has been selected before performing a function.
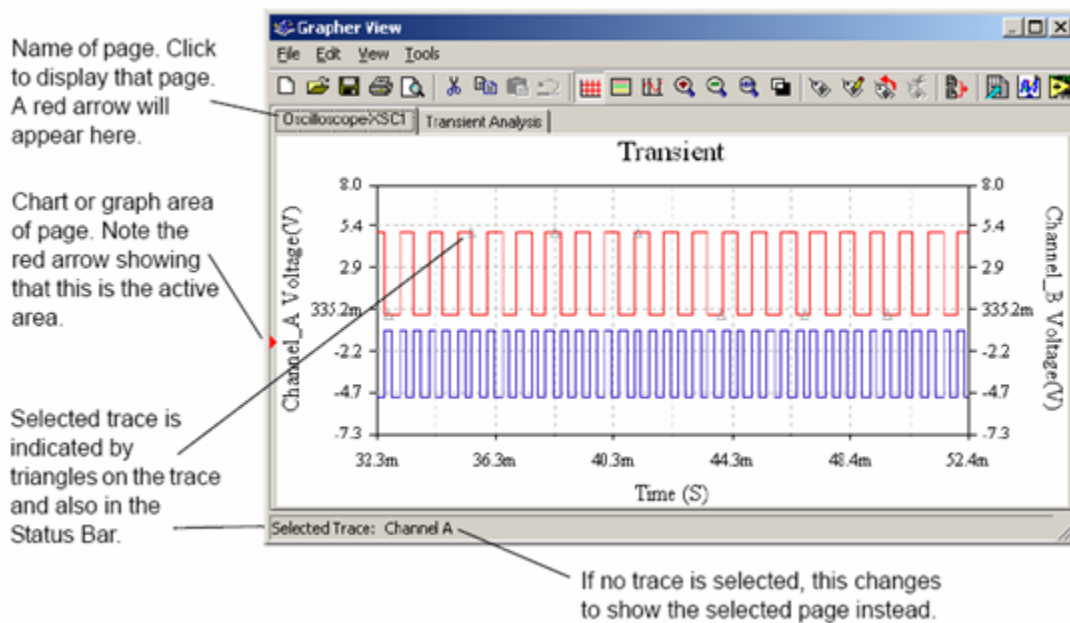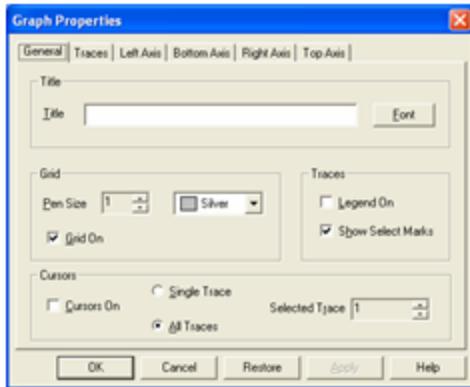
The Grapher

Through the properties pages, the Grapher provides extensive customization. Axes scales, ranges, titles, colors, line styles, and many more options can be modified. To access the page or standard properties pages, click Edit/Page Propertiesor Edit/Properties (Figure 4 and Figure 5).
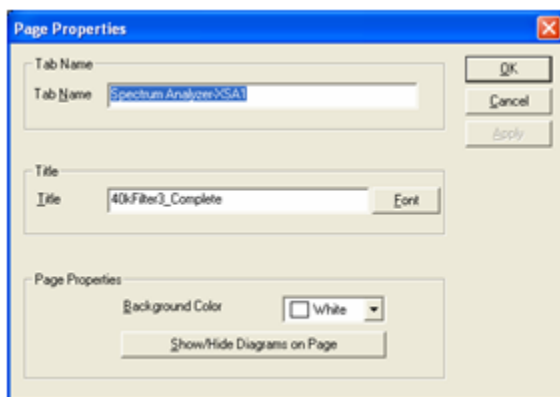
Grapher Page Properties

Graph Properties
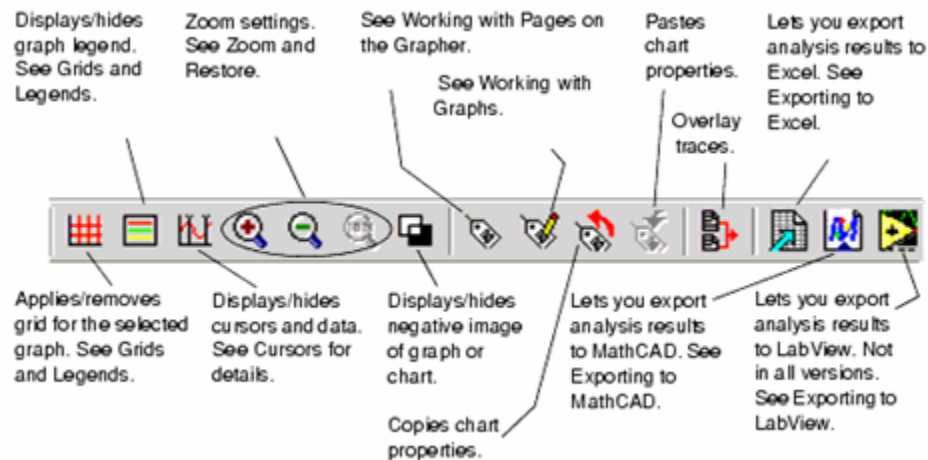
Cursors can be moved by clicking and dragging them with the mouse. Additionally, right-clicking on a cursor, will display cursor movement options. Users can move the cursor to a particular X-Value, Y-Value, and local Maxima or Minima in either direction (Figure 6). Cursors, legends and graph lines can be toggled on or off by clicking the corresponding toolbar buttons (Figure 7).

Cursor

Movement
Options

Displays/hides graph legend. See Grids and Legends.

Zoom settings. See Zoom and Restore.

See Working with Pages on the Grapher.

See Working with Graphs.

Pastes chart properties.

Overlay traces.

Lets you export analysis results to Excel. See Exporting to Excel.

Applies/removes grid for the selected graph. See Grids and Legends.

Displays/hides cursors and data. See Cursors for details.

Displays/hides negative image of graph or chart.

Lets you export analysis results to MathCAD. See Exporting to MathCAD.

Lets you export analysis results to LabView. Not in all versions. See Exporting to LabView.

Copies chart properties.

Grapher Toolbar

Results can also be exported to NI LabVIEW, Excel, or MathCAD. Graph data can also be saved to a variety of formats including NI LabVIEW Data (either .LVM or .TDM), comma separated values (.CSV), and plain text. To save results from the Grapher select File/Save As, and choose the desired file format.

Exercise: Analysis of Circuit Simulations in National Instruments Multisim

Exercise: Working with Analyses in Circuit Simulation

Approximate time to complete: 35 minutes

In this exercise, users will further explore the characteristics of the bandpass filter using analyses. Users will use AC, Transient, Fourier, and Monte Carlo Analyses and learn about analyses settings and how to configure the Grapher.

Objectives

- Compare AC Analysis to a Bode plot
- Compare Transient Analysis with the Oscilloscope
- Use expressions in analyses
- Understand how to set up and run Fourier Analysis
- Understand how to set up tolerances and run a Monte Carlo Analysis
- Learn how to format the output in the Grapher

Procedure

1. Load circuit [40kFilter3.ms9.](#) Notice that a load resistance has been added to the circuit (Rload) on the output of the filter. This is to facilitate an output power analysis.
2. Run the Simulation to get the Bode plotter and oscilloscope traces. Double-click the Bode plotter and Oscilloscope icons to open the instrument panels. Start the simulation by clicking on the lightning bolt button, or by pressing F5. Stop the simulation after the Bode plot is displayed. Close the instrument panels by clicking on the Close button on each instrument. Note:You can also open and close instrument panels by double-clicking on the instrument icons.

3. Open the settings for an AC Analysis (Simulate/Analyses/AC Analysis).

   1. On the Output tab, remove all variables from the Selected variables for analysis column on the right hand side of the dialog box. To do this, select all variables in the column and then click Remove.
   2. Select the $output variable and click Add.
   3. The test point will move to the right side under Selected Variables for Analysis.

4. Verify the output parameters and simulate.
5. Click Simulate.

The Grapher appears with multiple tabs. The last three include: one for the oscilloscope, one for the Bode plotter, and one for the AC Analysis. Compare the AC Analysis and the Bode Plotter graphs.

1. Perform the steps below to adjust the graph properties of the AC Analysis. These general steps can be used for adjusting any graph.
2. Left-click on the Magnitude graph (top graph) to make it active.

A small arrow on the left side of the window indicates which graph is active.

1. Right-click on the left axis to access the Graph Properties option.

   1. Select the Left Axis tab.
   2. Enter the following information in the Left Axis tab:
   3. In the Scale box, choose Decibels.
   4. In the Label dialog, type Gain (dB).
   5. In the Axis box, choose Enabled and a Pen Size of 1.
   6. In the Range box, choose a minimum range of –50 and a maximum range of 10.
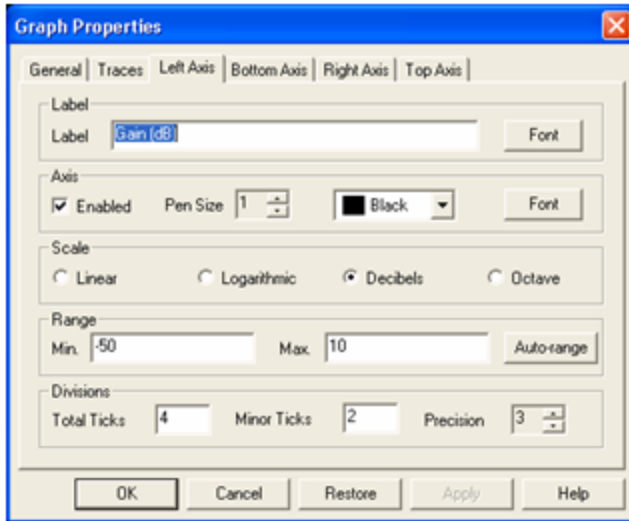
Setting the Graph Properties for the
Magnitude

1.   1.   1. In the Divisions section, choose Total Ticks of 4, Minor
Ticks of 2 and Precision of 3.
2. Click the Apply button.

2. Select the Bottom Axis tab.

1. Select Logarithmic Scale. In the Frequency Range, type
1000 for minimum and 1000000 for maximum.
2. Click Apply, then OK.

2. Adjust the bottom (Phase) graph by setting the Graph Properties as
shown in the following figure. Then click on the Bottom Axis tab and
change the range to 1000 to 1000000.
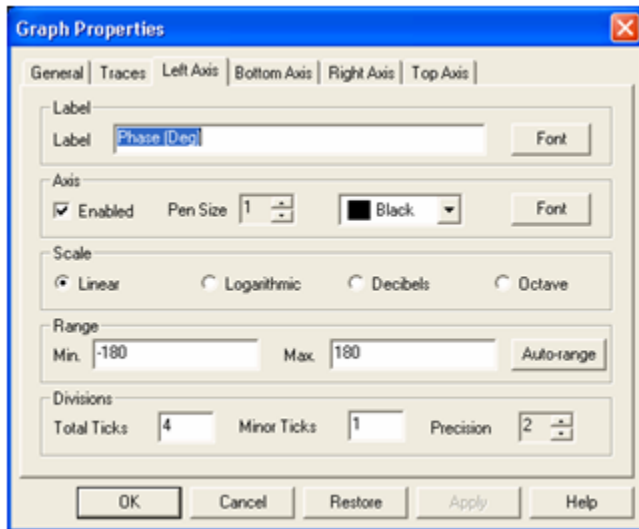
Setting the Graph Properties for the
Phase

Now compare the outputs of the Bode plot and the AC analysis by directly overlaying the magnitude traces.

1. Left-click on the magnitude plot of the Bode plot to select it.

    1. Select Overlay Traces from the Tools menu.
    2. Pick Graph_1 of the AC Analysis (this should be the second last entry in the list).

A new page will open in the Grapher that displays the two traces overlaid.

You can zoom in to see the region where both traces exist. Press and hold the left mouse button as you drag a box around the peak of the graph.
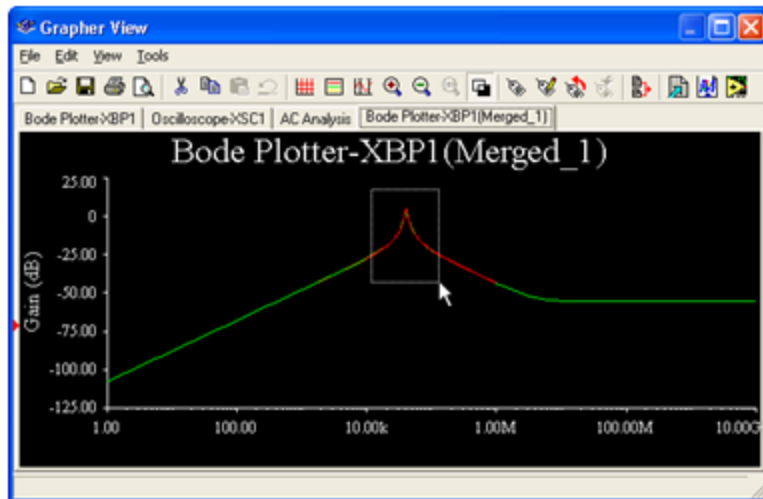
Zooming in on Overlaid Traces

Notice the results are slightly different. This is because the two methods used different sampling rates. You can control the sampling rate when you set up the instrument or the analysis.

1. Investigate how to make precise measurements in the Grapher.

    1. Select the Bode Plot Tab in the Grapher.
    2. Turn on the cursors by selecting Show/Hide Cursors from the Viewmenu.
    3. Select one of the cursors and right-click on it.
    4. Select Go to next Y_MAX to find the peak.
    5. Select Set Y_Value =>and enter the value that is 3 less than the max. This will give you one of the -3dB points.
    6. Read the resulting value in the numeric display box.

2. Perform Transient Analysis (Simulate/Analyses/Transient Analysis).

    1. Set the analysis settings as shown below. Note: You may need to expand the dialog box by clicking the More button.
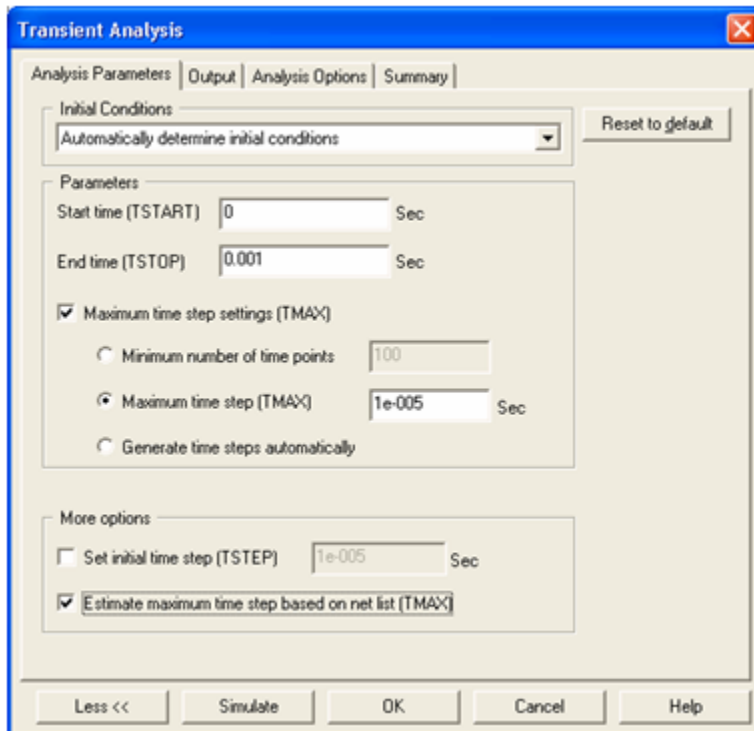
Setting Up the Transient Analysis

1. 1. Switch to the Outputtab.
   2. Select test point nodes $input and $output as the Selected Variables for Analysis.
   3. Click the Simulate button. Compare the graph with the oscilloscope display.

You will now set up to run a Fourier analysis.

1. Open the instrument panel of the function generator and change the waveform to a square wave.

   1. Choose Simulate/Analyses/Fourier Analysis.
   2. Select both Estimate buttons to set the sampling options and transient Analysis options automatically.
   3. Switch to the Outputvariablestab.
   4. Select $input and $output as the Selected Variables for Analysis.

5. Click Simulate.

Note the output is presented on two separate pages in the Grapher.

Optional Section (time permitting):

Perform a Monte Carlo Analysis, (Simulate/Analyses/Monte Carlo).

1. From the front dialog box, select Add a new tolerance function.

    1. In Parameter Type, choose Device Parameter.
    2. In Device Type select Capacitor. Choose cc1.
    3. In the Tolerance section, select Percent. For Tolerance Type, assign 20 and click Accept.
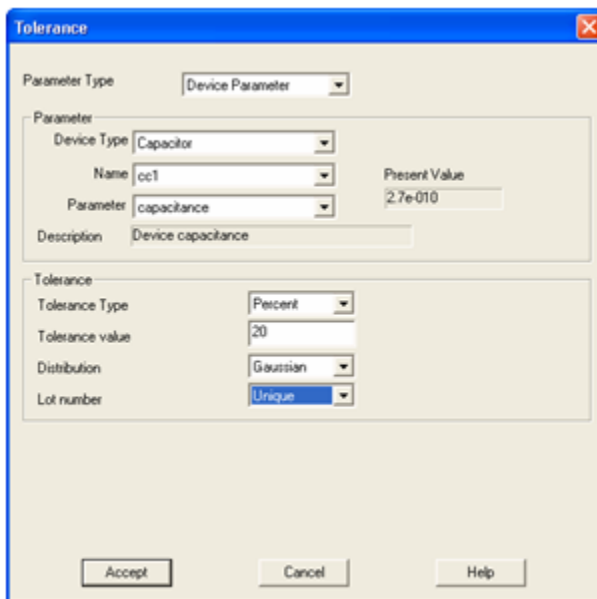
Setting Device Tolerances

1.   1. Select Add a new tolerance again, and repeat the above procedure. Select cc2 this time.
      2. Now configure the Analysis Parameters.

3. Click on the Analysis Parameters tab.
4. Set it up to run an AC Analysis, with 5 runs, and set the Output variable to $output.
5. Click on the Edit Analysis button and then configure the settings for the AC analysis.
6. Set the FSTART to 1 kHz, the FSTOP to 1 MHz, the Number of points per decade to 100 and the Vertical scale to Decibel.
7. Click the Simulate button.

 SOLUTION

Integrated Design with Other National Instruments Products

LabVIEW

National Instruments LabVIEW is a graphical programming environment that can be used to automate test and measurement functions, for the purpose of testing and validating designs. Multisim data saved in .LVM or .TDM format can be easily loaded into LabVIEW using express Virtual Instrument (VI) technology. Simulation data can then be overlaid on top of measured results to quickly and easily verify designs.

Loading .LVM and .TDM Data Files

To load Multisim simulated data that were saved in a .LVM or .TDM file, use the Read from Measurement File Express VI. This VI is located in the Programming/File IOpalette. For help on this file, consult the LabVIEW help file. The Express VI can be configured to read either .LVM or .TDM files.
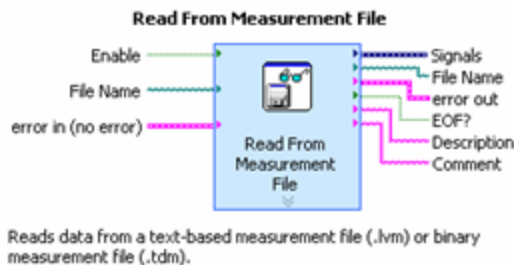
Read from Measurement
File Express VI

Note: The following only applies to .LVM files.The EOF? output terminal of the Express VI indicates true if there is more data contained in the .LVM file. This value is especially useful when reading data that were saved from a Bode Analysis, or from any plot window with more than one graph display. Simply place the Read From Measurement File Express VI into a while loop, with the EOF? output terminal wired to the exit condition terminal.

Note: The following only applies to .TDM files. To load .TDM files into LabVIEW which contain multiple graphs, lower level storage express VIs are required. Figure 2 below gives a demonstration of how one might load multiple graphs from a .TDM file.
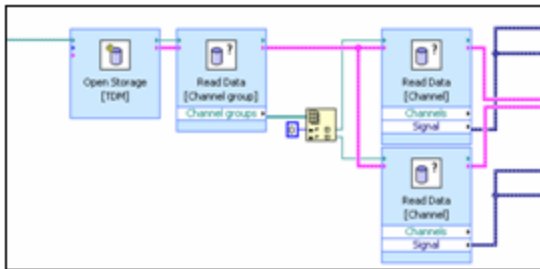
Loading Multiple Graphs
from a .TDM File

LabVIEW Based Virtual Instruments in Multisim

For further information on LabVIEW based Virtual Instruments in Multisim, consult the LabVIEW Virtual Instruments text in Section II.
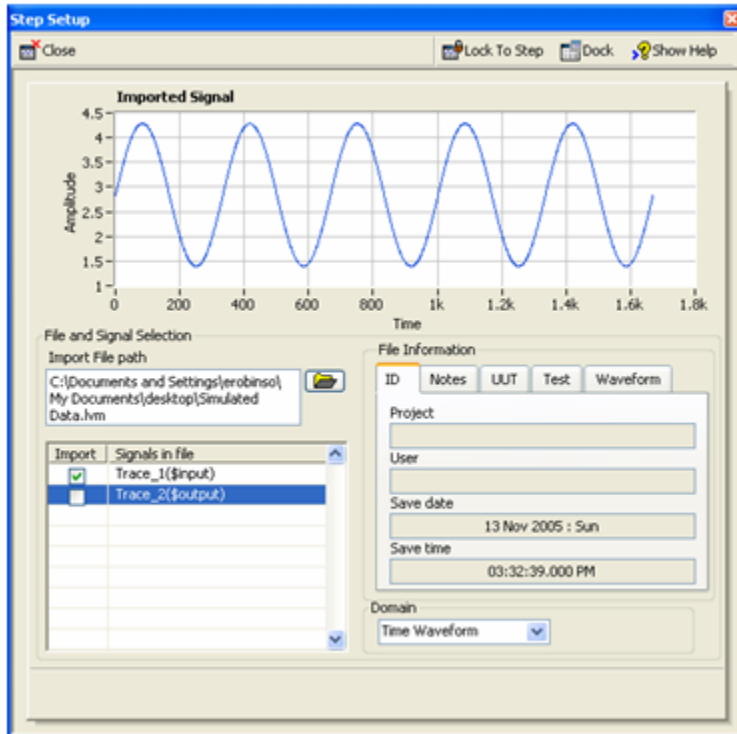
SignalExpress

SignalExpress introduces an innovative approach to configuring your measurements using intuitive drag-and-drop steps that do not require code development. Unlike traditional benchtop measurement tools, SignalExpress combines the optimal balance of measurement functionality and ease-of-use to assist designers in streamlining a variety of applications:

- Design modeling.
- Design verification.
- Design characterization.
- Device validation.
- Automated test troubleshooting.

Loading Multisim Data into SignalExpress

To load simulated data into SignalExpress, simply use the Load from LVM step, select Add Step/Analog/Load and save Signals/Load from LVM. Choose the file name to load, and select the traces to import. Use the Domain field to choose from either Time, or Frequency data. Click close and run the SignalExpress workbench file to load the Multisim data.

SignalExpress Load from LVM Step

ELVIS

National Instruments ELVIS is an ideal companion to any electronics lab that uses Multisim. ELVIS provides a breadboard prototyping environment, with built-in instruments including a Function Generator, Digital Multimeter (DMM), Oscilloscope, and Variable Power Supplies. The breadboard is detachable, allowing students to work on their projects and labs independently of the ELVIS unit.

ELVIS provides LabVIEW-based software for interacting with the Virtual Instruments. These instruments can be modified to load Multisim data for rapid comparison of simulated and measured data.
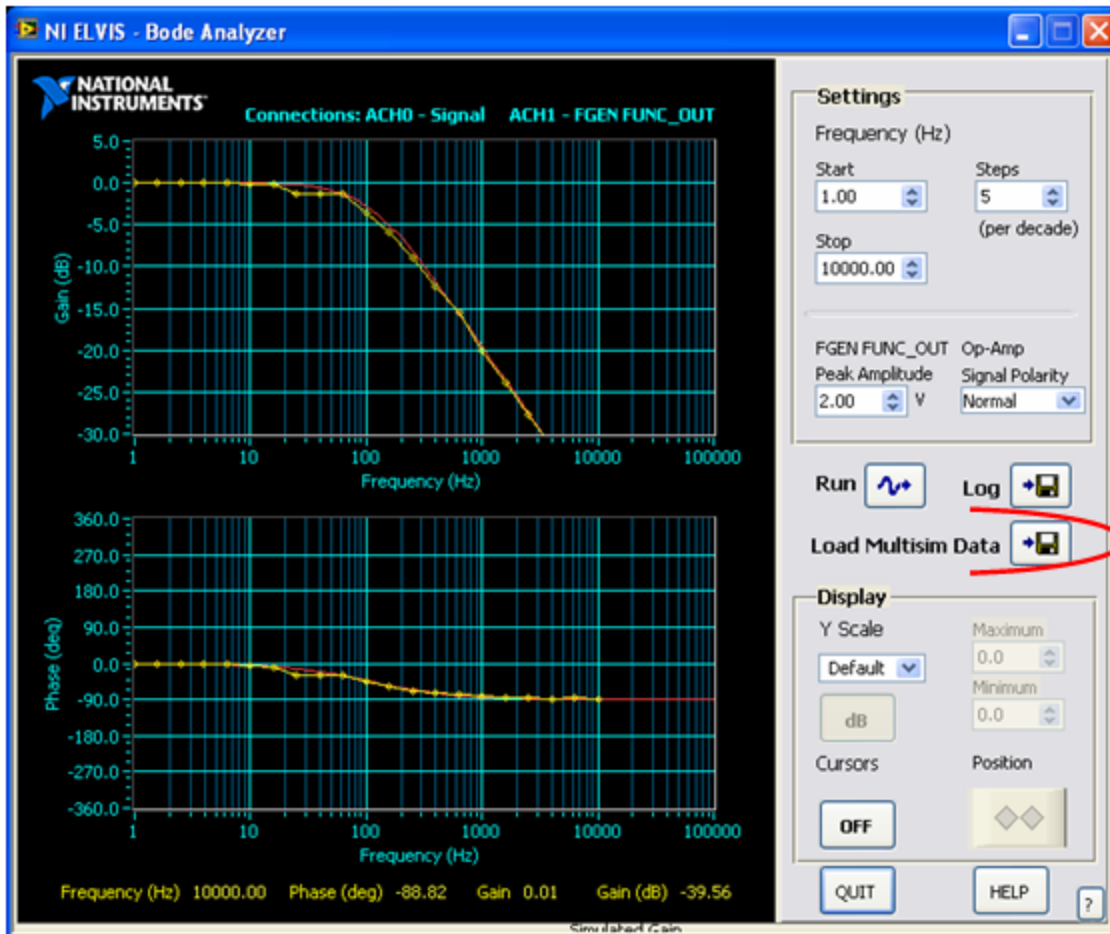
NI ELVIS Bode Analysis Comparing MultiSim Data

Enhanced Educational Features for Teaching with National Instruments Multisim

# Enhanced Features for Teaching

Overview

Multisim is used in colleges and universities around the globe. It has become an essential academic tool. Academic features available in Multisim provide students with intuitive methods for easily exploring circuit theory.

Because National Instruments products can be applied at the introductory, advanced and professional levels, students' proficiency with the software will naturally grow over time, allowing them to focus on tackling increasingly complex content and projects, not on learning new tools over and over as their needs increase.

Prototyping

3D Virtual Components

Multisim includes a set of components specifically designed for students with little to no experience working with parts in real life. When placed on the schematic, these 3D virtual components appear exactly as they look in the real world. These components are available in the Basic group of the Master Database, under the 3D_VIRTUAL family.
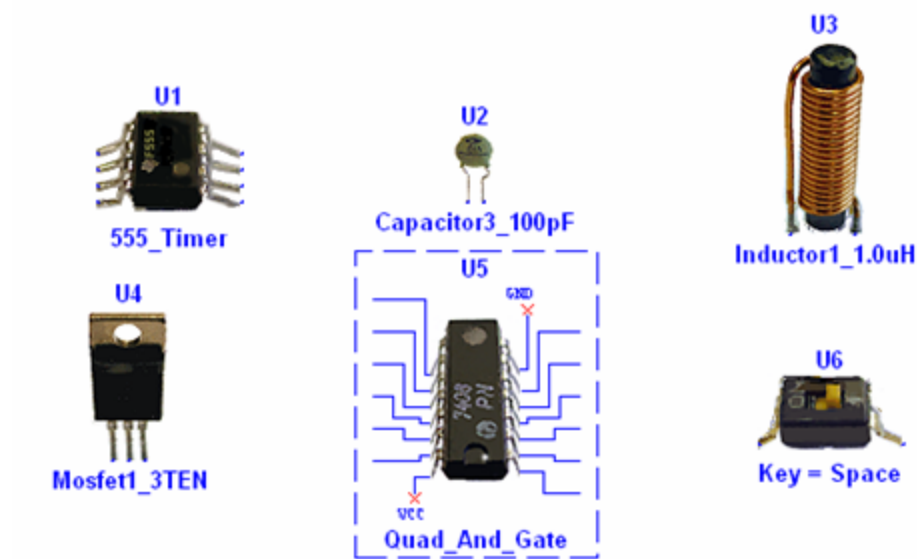
3D Virtual Components

Animated Components

Animated components provide students with visual feedback on circuit operation during simulation. Examples of animated components include LEDs, 7-Segment displays, and other interactive components such as switches and potentiometers. Animated components are located throughout Multisim's databases.

Rated Components

Rated Componentshave tolerances just like real-world parts. Users can control the tolerance level by editing the part properties. If the power being dissipated by a virtual rated resistor is above the threshold, the resistor will "explode" on the schematic, and act as an open circuit. Rated Virtualcomponents can be found in the RATED_VIRTUAL family of the Basic group in the Master Database.

3D Virtual Breadboard

Multisim also includes a 3D Virtual Breadboard. The breadboard can be used to teach students important prototyping techniques. Nets and components will turn green on the schematic when their corresponding breadboard elements have been correctly wired. The breadboard also includes a Design Rules Check (DRC) and Connectivity Check.
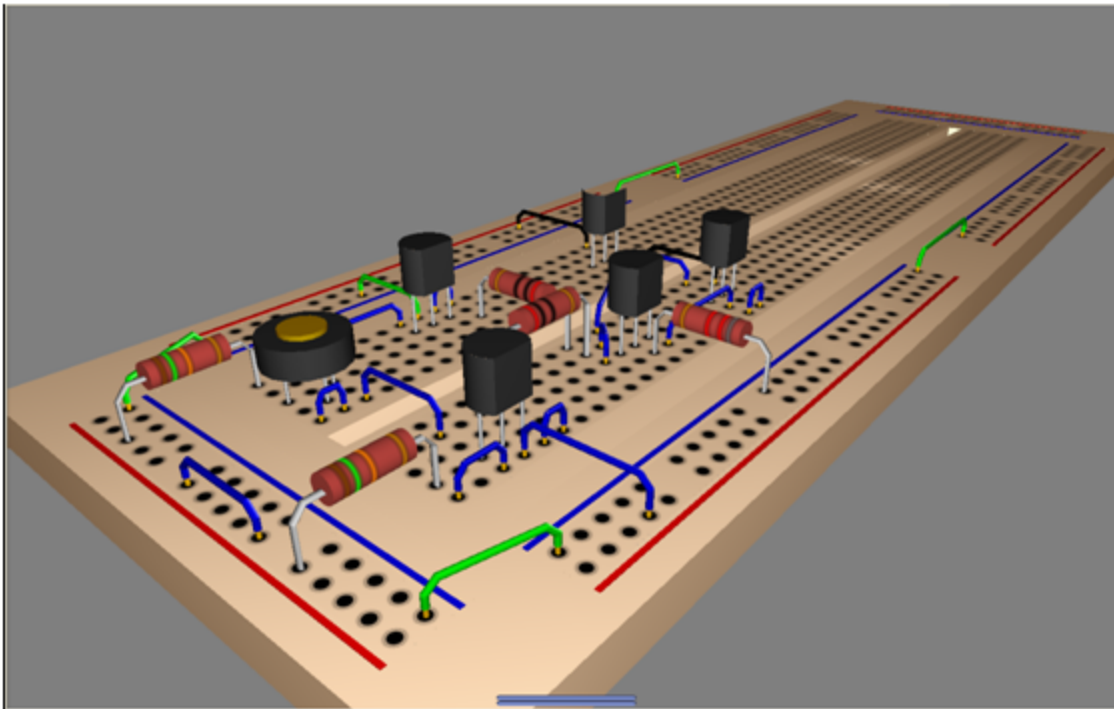

3D Virtual Breadboard

3D Virtual ELVIS

In addition to the standard 3D Virtual Breadboard, students also have the option to create NI ELVIS schematics, which will have a corresponding 3D ELVIS representation. The virtual instruments included with NI ELVIS such as the function generator, oscilloscope, and variable power supplies

have also been modeled. To create an NI ELVIS schematic, click File/New/ELVIS Schematic.

To access the ELVIS virtual instruments, double-click on the terminals corresponding to the desired instrument.
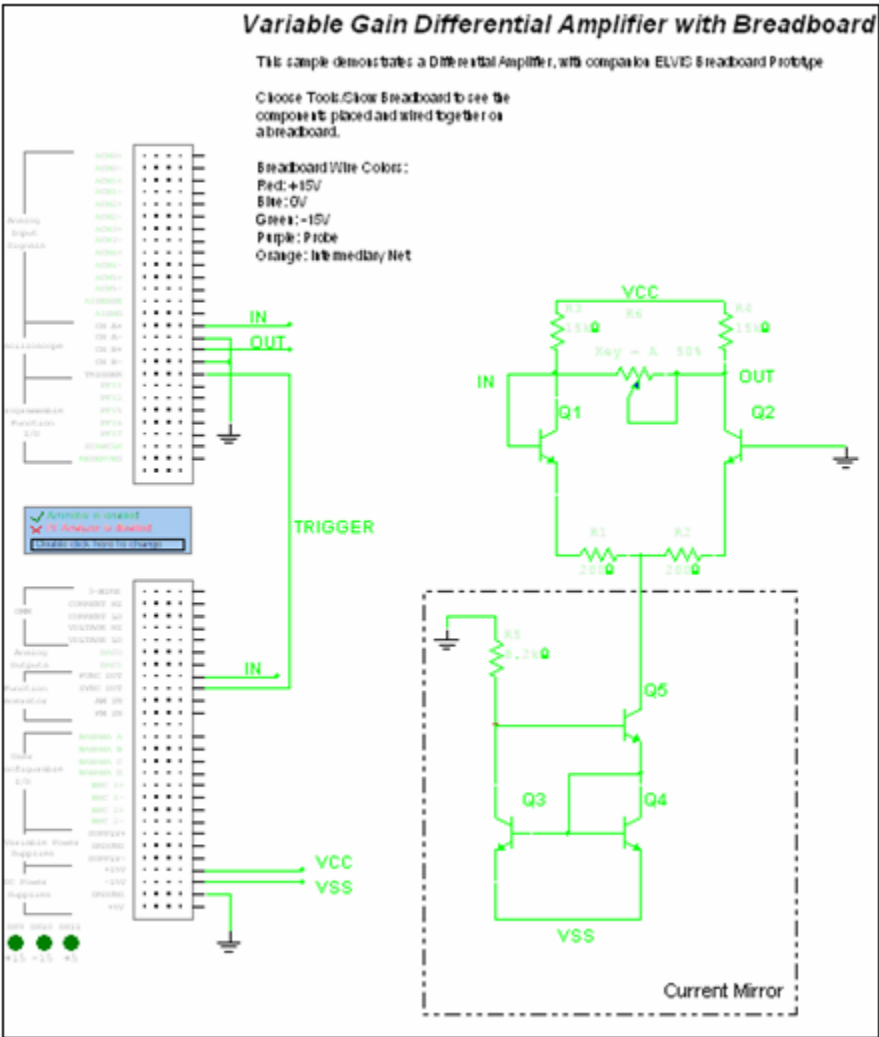

3D NI ELVIS Schematic

3D Virtual NI ELVIS

Troubleshooting

Circuit Faults

Instructors have the ability to insert faults into schematics, and have students hone their troubleshooting skills by trying to find the faults. The injected faults can be hidden to force students to apply fundamental techniques of troubleshooting. To add a fault into a component, double-click on the component, and select the Faults tab.

Faults include open circuits on a particular terminal, short circuits between two pins of a component, and resistive losses measured in Ohms.
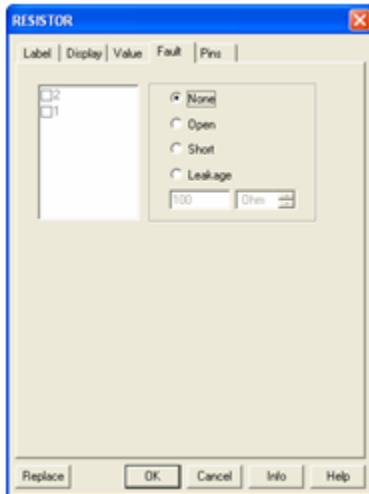
Adding Faults to
Components

Black Boxes

Instructors can also use subcircuits as black boxes, and engage students in
more sophisticated circuit analysis techniques. To create a black box
problem, use the information in Section I on creating subcircuits.
Note:Hierarchical blocks should not be used as black boxes because the
internal circuitry is stored in a separate file, which students would be able to
view.

Circuit Restrictions

Circuit restrictions play a very important role when creating black box
problems, or inserting circuit faults. Restrictions allow instructors to
prevent students from seeing subcircuits, hide the faults, and limit the
number or available components and instruments. In this manner, as an
example, it is possible to force students to use an oscilloscope to make gain

measurements when determining the bandwidths of filters, as a bode plotter or AC analysis may not always be available.

Global and circuit restrictions are available under the Options menu. Restrictions are locked by password.
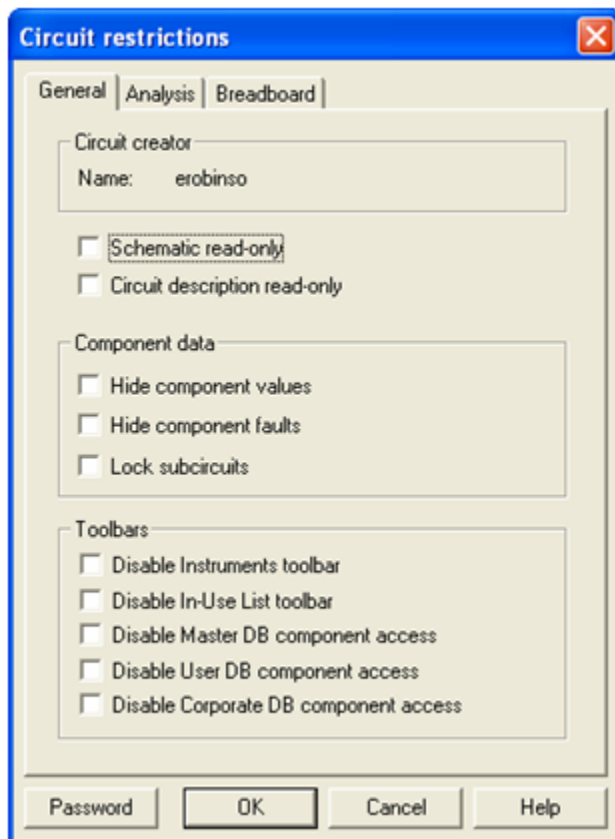
Circuit Restrictions Dialog Box